

Copy No. _____



Defence Research and
Development Canada

Recherche et développement
pour la défense Canada



MASTSAS – Mast Structural Analysis Software

Reference Manual

*T. S. Koko
D. P. Brennan
M. E. Norwood
T. MacAdam
P. A. Rushton*

*Martec Limited
Suite 400, 1888 Brunswick St.
Halifax, NS B3J 3J8*

Contract Number: W7707-008062/001/HAL

Contract Scientific Authority: Dr. M. J. Smith, 902-426-3100 x383

Terms of Release: *The information contained herein is proprietary to Her Majesty and is provided to the recipient on the understanding that it will be used for information and evaluation purposes only. Any commercial use including use for manufacture is prohibited. Release to third parties of this publication or information contained herein is prohibited without the prior written consent of Defence R&D Canada.*

Defence R&D Canada – Atlantic

Contract Report

DRDC Atlantic CR 2004-088

April 2004

Canada

This page intentionally left blank.

MASTSAS – Mast Structural Analysis Software

Reference Manual

T. S. Koko
D. P. Brennan
M. E. Norwood
T. MacAdam
P. A. Rushton

Martec Limited
Suite 400, 1888 Brunswick Street
Halifax, NS
B3J 3J8

Contract number: W7707-008062/001/HAL

Contract Scientific Authority: Dr. M. J. Smith, (902) 426-3100 x383

Terms of Release: The information contained herein is proprietary to Her Majesty and is provided to the recipient on the understanding that it will be used for information and evaluation purposes only. Any commercial use including use for manufacture is prohibited. Release to third parties of this publication or information contained herein is prohibited without the prior written consent of Defence R&D Canada.

Defence R&D Canada – Atlantic

Contract Report

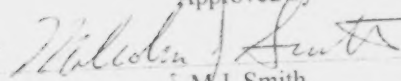
DRDC Atlantic CR 2004-088

April 2004

Author

T.S. Koko

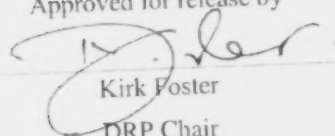
Approved by



M.J. Smith

Contract Scientific Authority

Approved for release by



Kirk Foster

DRP Chair

The scientific or technical validity of this Contractor Report is entirely the responsibility of Martec Limited and the contents do not necessarily have the approval or endorsement of DRDC.

- © Her Majesty the Queen as represented by the Minister of National Defence, 2004
- © Sa majesté la reine, représentée par le ministre de la Défense nationale, 2004

Abstract

MASTSAS (MAST Structural Analysis Software) is a special-purpose, PC based computer program for rapid finite element modeling of warship mast structures. The MASTSAS software was developed under contract to DRDC Atlantic, using its HOOD (Hierarchical Object Oriented Database) toolkit. This Reference Manual, one of a series of three manuals documenting the MASTSAS software, provides the theoretical foundations and details on the capabilities offered by the program.

Résumé

MASTSAS (MAST Structural Analysis Software ou logiciel d'analyse des structures de mâts) est un programme spécialisé pour PC, destiné à la modélisation rapide par la méthode des éléments finis des structures des mâts des navires de guerre. Le logiciel MASTSAS a été développé par un entrepreneur lié par contrat à RDDC Atlantique, au moyen de la boîte à outils HOOD (Hierarchical Object Oriented Database ou base de données hiérarchique orientée objet) de cet organisme. Ce Guide de référence, qui fait partie d'une série de trois manuels qui documentent le logiciel MASTSAS, présente les bases théoriques ainsi que des détails sur les capacités offertes par ce programme.

This page intentionally left blank.

Executive summary

Introduction

Under the FELEX (Frigate Equipment Life Extension) project, a redesign of the HALIFAX class main mast is being considered for its mid-life refit. Since 1999, DRDC Atlantic has been developing capabilities for efficient modelling of warship mast structures, principally through the development of the MASTSAS software, so as to aid the engineering analysis of new or existing mast designs.

Principal Results

The MASTSAS software provides capabilities for efficiently modeling mast structures typical of warships. These include lattice masts (currently used on CF vessels), and enclosed, or plated, mast designs. Metallic and composite structural materials are supported. The report provides a Reference Manual for the MASTSAS software, describing the theoretical foundations and details of the capabilities offered by the program

Significance of Results

Engineering assessments of new or existing mast designs can be performed with MASTSAS in an efficient manner. The tools provided in MASTSAS and the design features of the software allow changes in the design to be assessed more rapidly than with a generic FE modelling package.

Future Plans

The development of the software was completed in 2003. No further development is planned at this time, although some modification may be required for the FELEX project.

Koko, T. S., Brennan, D. P., Norwood, M. E., MacAdam, T., and Rushton, P. A. (2004). MASTSAS – Mast Structural Analysis Software. Reference Manual. DRDC Atlantic CR 2004-088. Defence R&D Canada – Atlantic.

Sommaire

Introduction

Dans le cadre du projet FELEX (Frigate Equipment Life Extension ou prolongation de la vie de l'équipement des frégates), on envisage la réingénierie du grand mât des navires de la classe HALIFAX, pour leur remise à niveau de mi-vie. Depuis 1999, RDDC Atlantique élabore des capacités permettant de modéliser de façon efficace les structures des mâts des navires de guerre, principalement grâce au développement du logiciel MASTSAS, afin de faciliter l'analyse technique des nouveaux modèles de mât ou des modèles de mât existants.

Principaux résultats

Le logiciel MASTSAS est doté de fonctionnalités qui permettent de modéliser de façon efficace les structures de mât que l'on retrouve sur les navires de guerre. Ces structures comprennent les mâts-treillis (utilisés actuellement sur les navires des FC), ainsi que des modèles de mât à structure fermée ou composée. Ce logiciel prend en charge les structures en matériau métallique ou composite. Le rapport présente un guide de référence du logiciel MASTSAS, qui décrit les bases théoriques et les détails des capacités offertes par ce programme.

Portée

MASTSAS permet d'effectuer une évaluation technique efficace des nouveaux modèles de mât ainsi que des modèles existants. Les outils intégrés à MASTSAS ainsi que les fonctions de conception du logiciel permettent d'évaluer les changements apportés à la conception, plus rapidement qu'avec un logiciel de modélisation par éléments finis général.

Futures recherches

Le développement du logiciel a été achevé en 2003. Pour le moment, on ne prévoit pas d'autre travail de développement, bien qu'il se peut que certaines modifications doivent être apportées pour le projet FELEX.

Koko, T. S., Brennan, D. P., Norwood, M. E., MacAdam, T., et Rushton, P. A. (2004). MASTSAS – Mast Structural Analysis Software. Reference Manual. RDDC Atlantique CR 2004-088. R & D pour la défense Canada – Atlantique.

Table of contents

Abstract	i
Résumé.....	i
Executive summary.....	iii
Sommaire	iv
Table of contents.....	v
List of figures.....	vii
1. INTRODUCTION	1
1.1 Background	1
1.2 MASTSAS Documentation.....	1
1.3 Organization of This Manual	1
2. STRUCTURE OF MASTSAS SOFTWARE	3
3. MATERIAL AND SECTION PROPERTY DATABASES	6
3.1 Materials Database	6
3.1.1 Metals.....	6
3.1.2 Composites.....	6
3.1.3 Core Materials.....	7
3.2 Section Properties.....	8
3.2.1 Beam Properties	8
3.2.2 Plate/Shell Properties	8
3.2.3 Composite Laminate Lay-up Sequences	8
4. GEOMETRIC MODELING.....	9
4.1 Modelling Philosophy	9
4.2 Mast Object	9
4.3 Cross Sections and Cross Section Assemblies	10
4.4 Mast Brace End Points	14
4.5 User-Generated End Points	15

4.6	Braces	16
4.7	Bays and Bay Generators	18
4.8	Gusset Plates	23
4.9	Multisided Stiffened Panels and Cutouts	28
4.10	Mast Geometry Modelling Wizards	32
4.10.1	Cross Section Assembly Wizard	32
4.10.2	Brace Wizard.....	40
4.10.3	Brace End Point Wizard	40
5.	EQUIPMENT	41
6.	BOUNDARY CONDITIONS	44
7.	LOADS	46
7.1	Load Case Manager.....	46
7.2	Self Weight and Live Loads.....	51
7.3	Loads Due to Sea Generated Ship Motion	51
7.4	Wind Loads	53
7.5	Air Blast Loads.....	53
7.5.1	Blast Speed and Over Pressure.....	54
7.5.2	Dynamic Pressure.....	55
7.5.3	Diffraction Phase Pressure Loading	55
7.5.4	Drag Loading.....	56
7.5.5	Interference Effects	57
7.5.6	Temporal Variation of Blast Load for lattice Masts.....	57
7.5.7	Temporal Variation of Blast Load for Enclosed Masts.....	59
7.6	Underwater Shock Loads (USL)	59
7.6.1	Shock Spectrum Method (SSM).....	60
7.6.2	Base Acceleration Method (BAM).....	60
8.	FINITE ELEMENT MESH AND ANALYSIS SETUP	61
9.	POST-PROCESSING.....	64
9.1	Strength and Stability Adequacy Checks	64
9.2	Vortex Shedding Checks	67
10.	REFERENCES	68

List of figures

Figure 2.1: Scene Graph Representation of a Main Trunk.....	5
Figure 4.1: 'MastObject' and Its Children.....	9
Figure 4.2: GUI for the Base Cross Section Class	10
Figure 4.3: Modifying the Local Coordinate System	11
Figure 4.4: Cross Section GUI	11
Figure 4.5: Representation of CrossSectionAssemblies in the Scenegraph Tree Control.....	12
Figure 4.6: Contents/Structure of a Typical CrossSectionAssembly	13
Figure 4.7: CrossSectionAssembly Property Page (GUI)	14
Figure 4.8: Creating a Brace Endpoint.....	15
Figure 4.9: MastBraceEndPoint Property Page (GUI)	15
Figure 4.10: Creating a New Brace	16
Figure 4.11: Page 1 of Brace Generator Wizard (End Point Selection).....	16
Figure 4.12: Page 2 of Brace Generator Wizard (Brace Properties)	17
Figure 4.13: Page 3 of Brace Generator Wizard (Brace Orientation)	17
Figure 4.14: Brace Property Page (GUI).....	18
Figure 4.15: MASTSAS Bay.....	19
Figure 4.16: Hierarchy of Bay Generators within CrossSectionAssemblies.....	19
Figure 4.17: Lattice Mast Configuration Types	20
Figure 4.18: BayGenerator Property Page (GUI).....	21
Figure 4.19: Doubling the Number of Bays Using the BayGenerator GUI	21
Figure 4.20: Four Sided Mast Plate (Highlighted in Purple) Property Page	22
Figure 4.21: Four Sided Mast Plate Material Local Coordinate System.....	23
Figure 4.22: Gusset Plates Generated by MASTSAS Wizard.....	24
Figure 4.23: Gusset Plates Property Page.....	24

Figure 4.24: Gusset Plate Configurations: (a) GP1, (b) GP2, (c) GP3, (d) GP4	26
Figure 4.25: Details of Gusset Plate (Type GP7)	28
Figure 4.26: Adding Stiffened Panels to Cross-Sections	29
Figure 4.27: Mast Model Including New Stiffened Panel.....	29
Figure 4.28: Close-Up of Stiffened Panel	30
Figure 4.29: Multi-Sided Stiffened Panel Property Page	30
Figure 4.30: Defining Cutouts	31
Figure 4.31: MASTSAS-Generated FE Meshes with Cutouts	31
Figure 4.32: Creating a New Main Trunk (CrossSectionAssembly).....	32
Figure 4.33: Defining Overall Dimensions of the Main Trunk	33
Figure 4.34: Defining Cross-Sectional Properties	33
Figure 4.35: Defining the Bay Generators.....	34
Figure 4.36: Generating a CrossSectionAssembly with Multiple Bay Generator Types	34
Figure 4.37: Screen Captures of CrossSectionAssembly Wizard Dialog Boxes Used to Define Multiple Bay Generator Types.....	35
Figure 4.38: Main Trunk with Two Different Bay Types	35
Figure 4.39: Assigning Lattice Bay Generator Properties Using the Wizard.....	36
Figure 4.40: Assigning Enclosed Bay Generator Stiffener Orientation Using the Wizard	36
Figure 4.41: Assigning Enclosed Bay Generator Stiffener Properties Using the Wizard	37
Figure 4.42: Assigning Main Trunk Orientation Using the Wizard.....	38
Figure 4.43: Assigning Yardarm Orientation Using the Wizard.....	38
Figure 4.44: Assigning Yardarm Attachment Location Using the Wizard	39
Figure 4.45: Resulting Yardarm After Attachment with Main Trunk.....	39
Figure 5.1: Viewing the Contents of the Equipment Database	41
Figure 5.2: MastEquipmentCollector Property Page.....	42
Figure 5.3: Selecting the Type of Equipment to be Added to the Model.....	42

Figure 5.4: Defining the Equipment Attachment Location and Orientation	43
Figure 5.5: MastEquipmentObject Property Page.....	43
Figure 6.1: BoundaryConditionCaseManager Property Page	44
Figure 6.2: Model Showing Fixed Boundary Condition (Red Lines).....	45
Figure 6.3: Model Showing Pinned Boundary Condition (White Lines).....	45
Figure 7.1: LoadCaseManager Property Page.....	46
Figure 7.2: Initial Page of the LoadCase Wizard	47
Figure 7.3: Ship Motion Loads Wizard Page	47
Figure 7.4: Wind Loads Wizard Page	48
Figure 7.5: Air Blast Loads Wizard Page.....	48
Figure 7.6: Underwater Shock Loads Wizard Page	49
Figure 7.7: Live Loads Wizard Page	49
Figure 7.8: MASTSAS Display of Live Point Loads.....	50
Figure 7.9: Wind Load Post Processing Capabilities	51
Figure 7.10: Vertical Accelerations Amidships (DMEM 10)	52
Figure 7.11: Overpressure versus Range for 1-MT Weapon (Adapted from Biggs (1964))....	54
Figure 7.12: Overpressure and Dynamic Pressure Positive Durations Versus Range (1 MT Weapon) (Biggs 1964).....	55
Figure 7.13: Schematic Representation of Temporal Variation of Blast Load for Lattice Mast Members.....	58
Figure 7.14: Schematic Representation of Structural Regions Affected by Blast Loading	58
Figure 7.15: Blast Loading on Enclosed Masts (Lloyd's, 2000).....	59
Figure 8.1: Creating a New Boundary Detail	61
Figure 8.2: Default Mesh Settings GUI.....	61
Figure 8.3: Boundary Detail Property Page	62
Figure 8.4: Creating a DSA Database of the MASTSAS Model	62
Figure 8.5: Input Data for Analysis Setup.....	63

Figure 9.1: Flowchart of Algorithm to Determine the Mast Members Factor of Safety (Crocker et al, 1999)	65
Figure 9.2: Flowchart of Algorithm to Determine the Mast Members Factor of Safety (Crocker et al, 1999)	66

1. INTRODUCTION

1.1 Background

MASTSAS (acronym for MAST Structural Analysis Software) is a special-purpose, PC based computer program for rapid finite element modeling of masts. The MASTSAS software development is based on the Object Oriented Programming (OOP) technology. The system is built on the HOOD (Hierarchical Object Oriented Database) toolkit [1]. Thus all MASTSAS objects inherit from HOOD base classes and thus they inherit all the functionality of the base classes such as searching, I/O, setting privileges, as well as graphics functionality such as drawing and picking.

MASTSAS provides the following capabilities and features:

- Rapid geometric modeling of lattice, enclosed or combined lattice-enclosed masts;
- Rapid finite element modeling of the various types of masts;
- Interface with DSA-VAST and ANSYS finite element packages;
- Special-purpose post-processing to verify integrity of mast structures, based on strength, stability and vortex shedding criteria;
- Capabilities for defining and applying various types of loads (wind, blast sea load, self weight, and underwater shock) to mast structures;
- Capabilities for modeling masts made of metallic or composite materials;
- Provides a user friendly, PC based modeling environment

1.2 MASTSAS Documentation

The MASTSAS software documentation is presented in three volumes as follows:

- Volume 1 contains the User's Manual that provides guidance on the use of the software system
- Volume 2 (this volume) contains the Reference Manual that provides the theoretical foundations and details on the capabilities offered by the program.

Volume 3 contains the Examples Manual that provides sample problems/models generated by the MASTSAS system.

1.3 Organization of This Manual

The remainder of this manual is organized as follows:

- Chapter 2: Provides an overview of the MASTSAS software system. The hierarchical structure of the program as well as the symbols appearing in the program are highlighted.
- Chapter 3: Describes the materials and section properties databases. Highlights of the properties required for a MASTSAS analysis are provided.

- Chapter 4: Discusses the geometric modeling capabilities. The modeling philosophy and the methods for modeling various objects (mast components) are presented.
- Chapter 5: Presents details on the equipment database and the methods for applying equipment to the mast structure.
- Chapter 6: Discusses the methods for applying boundary conditions to the mast structure
- Chapter 7: Presents details on the various types of loads and methods for applying them to the mast structure
- Chapter 8: Provides methods of creating mast finite element models and setting up the finite element analysis data for various finite element analysis programs.
- Chapter 9: Discusses the MASTSAS post-processing capabilities in which details of the methods for verifying the integrity of mast structures are provided.

2. STRUCTURE OF MASTSAS SOFTWARE

In order to effectively use the MASTSAS software system it is important to develop an understanding of the scene graph concept. The MASTSAS scene graph is a hierarchical arrangement of objects (sometimes referred to as nodes) that collectively define the geometry and functionality of a mast model [1]. The hierarchical structure of the scene graph allows these objects to be grouped together in an object-oriented fashion in order to construct increasing more complex model components.

MASTSAS models will generally contain many types of objects (for a list of some of the more common MASTSAS objects, please refer to Table 2.1). However, these objects can be categorized into three main groups:

1. **The SceneGraph Object.**
The SceneGraph object is the topmost node in a MASTSAS (or HOOD) hierarchy. There is only one SceneGraph object in a given model.
2. **Parent Objects.**
Parent objects are the only objects that are allowed to have children. They serve to group other objects in order to logical collections. For example, a CrossSectionAssembly object is a parent object because it groups together a series of cross-sections and bay generators, which collectively, define the geometry of a mast trunk. Figure 2.1 provides the "scene graph" representation of this relationship, where the main trunk is depicted as a folder containing a series of sub-folders (each sub-folder, in turn, representing a child node).
3. **Child Objects.**
Child objects are used to represent more "primitive" objects, such as end-points or braces. In addition, child objects are also used in MASTSAS to represent customized functionality, such as launching a finite element analysis.

Table 2.1: List of Principal MASTSAS objects

Object Name	Description/Function
MastObject	Parent Object: Contains all Mast Objects
MastBraceEndPoint	Child Object: Represents the end-points of a brace
GeneratedBayPoint	Child Object: Represents the corner points of a bay
Brace	Parent Object: Represents a Mast Brace (stores end-points as child objects)
BayGenerator	Parent Object: Stores all information related to the geometry of a mast bay. Stores braces/and or stiffened panels as child objects

Table 2.1: List of Principal MASTSAS objects (cont.)

EnclosedBayGenerator	Parent Object (derived from BayGenerator). Used to represent enclosed bays.
LatticeBayGenerator	Parent Object (derived from BayGenerator). There are 8 derived types of latticeBayGenerators (A-H)
CrossSection	Parent Object: used to represent the cross-sections of a mast trunk, yardarm, or auxiliary trunk. May contain cut-outs and/or multi-sided stiffened panels as children
InterpolatedCrossSection	Parent Object (derived from crossSection). The corner points of an interpolated cross-section are automatically generated by interpolation of the corner points of neighboring cross-sections.
CrossSectionAssembly	Parent Object: used to represent the geometry of complex mast structural components. Stores cross-sections, bay generators, and gussets as children.
MainTrunk	Parent Object (derived from CrossSectionAssembly). Used to represent the mast main trunk.
Yardarm	Parent Object (derived from CrossSectionAssembly). Used to represent the mast yardarms.
AuxiliaryTrunk	Parent Object (derived from CrossSectionAssembly). Used to represent the mast auxiliary trunks.
MultiSidedStiffenedPanel	Parent Object: used to represent stiffened panels. May have brace objects (representing stiffeners) and end-points as children.
FourSidedStiffenedPanel	Parent Object: used to represent the sides of enclosed masts. May have brace objects (representing stiffeners) and end-points as children.
LoadCase	Child Object: used to store data related to mast specific load cases. There are six derived types of load cases developed for use in the MASTSAS code: self-weight, ship motion, wind load, blast, underwater shock, and live loads.
LoadCaseManager	Parent Object: used to manage all load cases.
BoundaryConstraintCase	Child Object: used to store data related to mast boundary conditions. Currently only point boundary conditions are available in MASTSAS.
BoundaryConstraintCaseManager	Parent Object: used to manage all boundary condition cases.

Table 2.1: List of Principal MASTSAS objects (cont.)

MastEquipment	Child Object: used to store all data related to mast equipment.
MastEquipmentCollector	Parent Object: used to manage all mast equipment objects.
FEBoundaryDetail	Parent Object: used to generate and manage all data related to finite element mesh generation.
MastMeshManager	Parent Object: used to store FEFBoundary Details
MastAnalysisType	Child Object: used to store all data related to the setup of a finite element solution.
Gusset	Child Object: used to store data used to define the location, orientation, and shape of a gusset plate. User's cannot generate gussets directly. Gussets are generated automatically by the code.

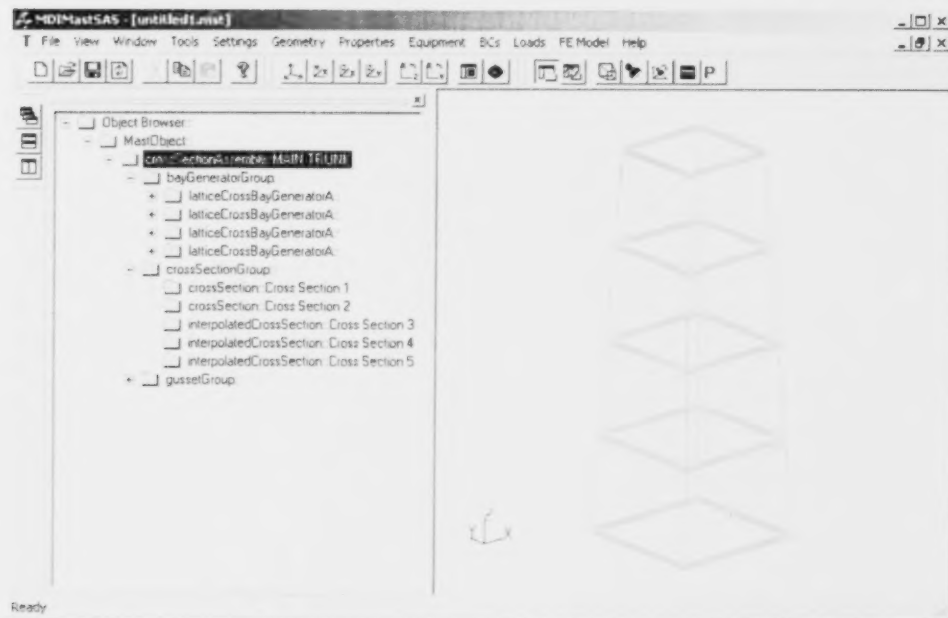


Figure 2.1: Scene Graph Representation of a Main Trunk

3. MATERIAL AND SECTION PROPERTY DATABASES

3.1 Materials Database

3.1.1 Metals

The metals database within MASTSAS contains a number of options, including various grades of aluminum and steel materials. The following properties may be defined for each material:

Elastic properties

- Young's modulus, E
- Poisson's ratio, ν
- Density, ρ
- Isotropic hardening modulus E_{PI}
- Kinematic hardening modulus E_{PK}

Strength properties:

- Yield strength, σ_{yield} and
- Ultimate stress, σ_{ult}

3.1.2 Composites

The composite materials database within MASTSAS contains various *grades* of glass-reinforced plastics (GRP), carbon fiber reinforced plastics (CFRP), and Kevlar fiber reinforced plastics (KFRP). Here, the term '*grade*' is used to distinguish between various instances involving the same material type and differences in fiber volume content, material sources, fabrication methods, etc. The following properties may be defined for each material:

Elastic Properties:

- Young's modulus in material longitudinal direction, E_{11}
- Young's modulus in material transverse direction E_{22}
- Young's modulus in material thickness direction, E_{33}
- Shear modulus in material 1-2 plane, G_{12}
- Shear modulus in material 2-3 plane, G_{23}
- Shear modulus in material 1-3 plane, G_{13}
- Poisson's ratio in material 1-2 plane, ν_{12}
- Poisson's ratio in material 2-3 plane, ν_{23} and
- Poisson's ratio in material 1-3 plane, ν_{13} .

Strength Properties:

- Longitudinal strength in tension, X_T
- Longitudinal strength in compression, X_C
- Transverse strength in tension Y_T
- Transverse strength in compression, Y_C
- Tensile strength in thickness direction, Z_T
- Compressive strength in thickness direction, Z_C
- Shear strength in 2-3 material principal plane, R
- Shear strength in 1-3 material principal plane, S
- Shear strength in 1-2 material principal plane, T

All strength properties refer to the principal material directions.

Strain Capacities:

- Longitudinal strain capacity in tension, X_{eT}
- Longitudinal strain capacity in compression, X_{eC}
- Transverse strain capacity in tension Y_{eT}
- Transverse strain capacity in compression, Y_{eC}
- Tensile strain capacity in thickness direction, Z_{eT}
- Compressive strain capacity in thickness direction, Z_{eC}
- Shear strain capacity in 2-3 material principal plane, R_e
- Shear strain capacity in 1-3 material principal plane, S_e
- Shear strain capacity in 1-2 material principal plane, T

All strain capacities refer to the principal material directions.

3.1.3 Core Materials

The core materials database is comprised of various *grades* of solid and honeycomb core materials. However, since the solid type materials are most likely to be used for mast applications (due to their superior acoustic properties compared to honeycomb core materials), MASTSAS currently provides properties for solid core materials only. Properties of honeycomb core material will be provided in future versions. The following properties may be defined for solid core materials:

Elastic and Physical Properties:

- Young's modulus, E ;
- Poisson's ratio, ν ; and
- Density, ρ

Strength properties:

- Yield strength, σ_{yield} ; and
- Ultimate strain, ϵ_{ult}

3.2 Section Properties

3.2.1 Beam Properties

The current beam property database contains I-, Tee-, angle- and solid rectangular (flat) cross sections for rolled metal components. This database has been expanded to include rolled hollow circular and rectangular sections that are commonly used for mast structures. In addition, the database has been further expanded to include prefabricated composite I-, solid- and hollow circular and rectangular sections. The following properties may be defined for each material:

- Areal dimensions (widths, depths, thicknesses, etc); or
- Sectional properties (moments of inertia, I_{xx} , I_{yy} , I_{zz} , J , etc)

3.2.2 Plate/Shell Properties

MASTSAS has the capability for defining plate/shell components made from single-layered *homogeneous* materials. The following properties may be defined:

- Plate/Shell uniform thickness

3.2.3 Composite Laminate Lay-up Sequences

MASTSAS also has the capability to define the lay-up sequences for plate/shell components made from multi-layered materials. The following properties may be defined:

- Number of layers;
- Material identity for each layer; and
- Ply orientation and thickness of each layer

If the laminate is symmetric, data is required for only one half of the laminate, and the program automatically generates the lay-up sequence for the entire laminate. Sandwich components may also be defined by specifying a lay-up sequence that includes the face and core materials.

4. GEOMETRIC MODELING

4.1 Modelling Philosophy

MASTSAS provides users with a general modeling tool for the rapid development of FE models of ship masts using an object-oriented approach. Users can develop sophisticated models by creating simple geometric '*objects*', defining how these objects are related to one another, and then assembling these objects into more and more complex components. The use of this relational geometric concept means that objects, after being modified, can notify related objects that changes have been made, allowing these objects to then modify their own properties in an appropriate way. Examples of how this relational modeling concept is used in MASTSAS, including a description of the principal MASTSAS modeling objects, are provided in the following sections.

4.2 Mast Object

All MASTSAS models require one (and only one) **MastObject**. The **MastObject** forms the basis of all MASTSAS models, and acts as a collector (or parent) for all data created and defined by the user. Initially the **MastObject** contains only three children (see Figure 4.1): the **LoadCaseManager** (stores load information), the **BoundaryConstraintCaseManager** (stores boundary condition data), and the **MastEquipmentCollector** (responsible for managing all equipment attached to the mast model).

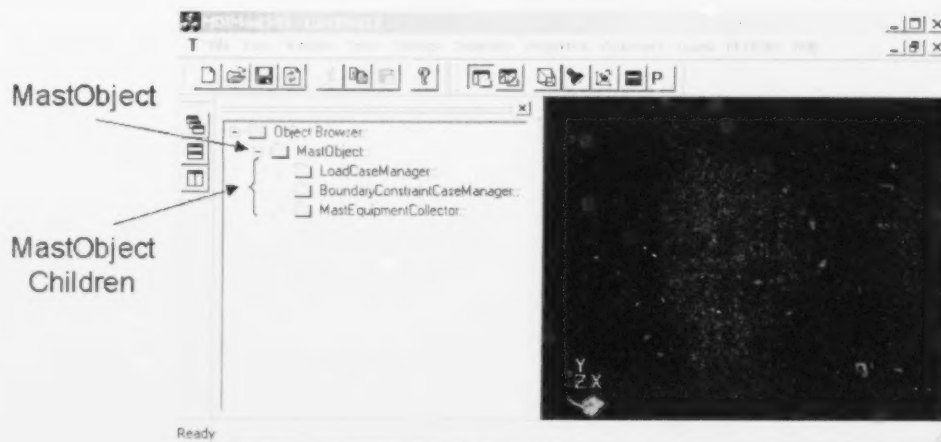


Figure 4.1: 'MastObject' and Its Children

4.3 Cross Sections and Cross Section Assemblies

The cross section is the lowest level geometric building block provided by the MASTSAS program. Cross sections are used to define the overall size and shape of masts and their appendages. In general, the task of creating cross sections is the responsibility of various MASTSAS wizards, and users cannot create cross sections directly.

The GUI developed for the base cross section class is shown below in Figure 4.2. Through this GUI, users can manipulate both the graphical representation of the cross section (i.e.: solid, wireframe, labeling of coordinates, etc.) and its size, shape, and orientation. It is important to note that MASTSAS cross sections are defined with respect to a local coordinate system and are assumed to be planar. For information purposes, the local coordinate system can be displayed by activating the "Plot Local Coordinate Axis" option provided in the interface (see Figure 4.2). The location and orientation of the local coordinate system is provided in the fields contained in the "Local Coordinate System" section of the interface. Manipulating these fields will modify the orientation of the cross section and any objects attached to it (see Figure 4.3 and Figure 4.4).

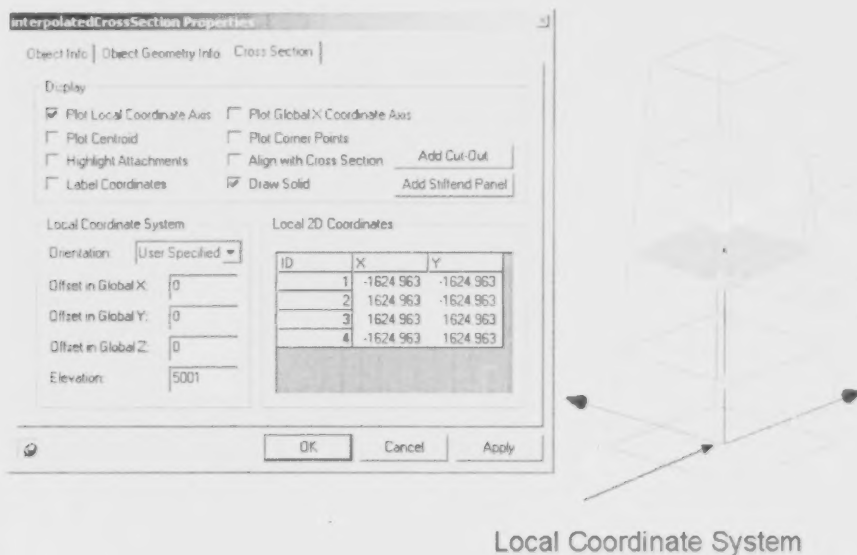


Figure 4.2: GUI for the Base Cross Section Class

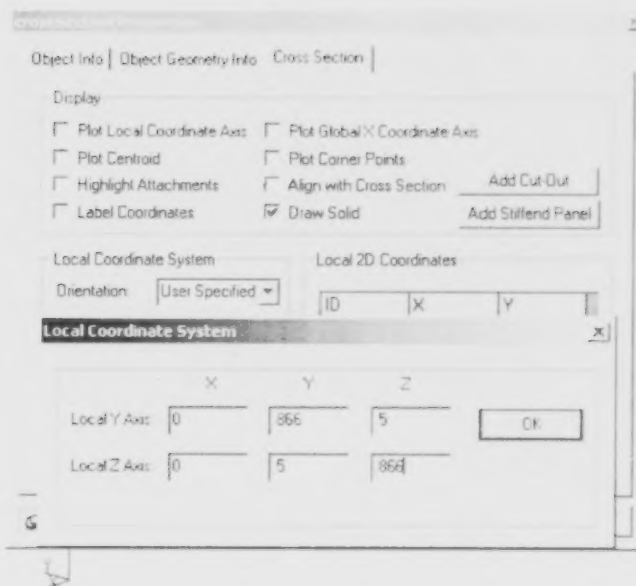


Figure 4.3: Modifying the Local Coordinate System

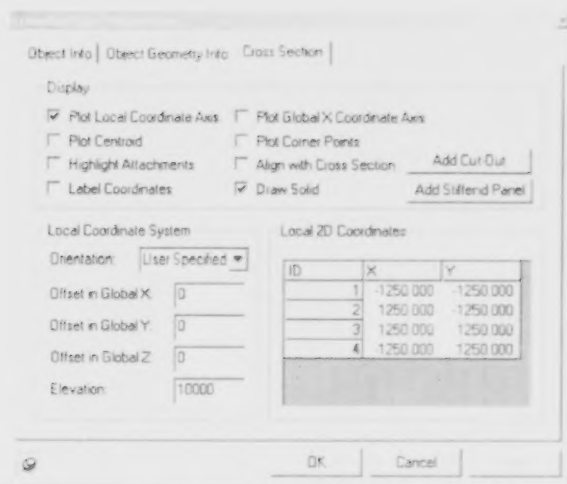


Figure 4.4: Cross Section GUI

There are three basic types of cross sections available, one general type and two special purpose types: the **interpolatedCrossSection** and the **bayAttachmentCrossSection**. The main distinction between the basic cross section object and the interpolated cross section class is that the size and location of the interpolated cross section is defined in terms of a pair of cross sections located above and below it. The **bayAttachmentCrossSection** feature provides specialized functionality for use in defining the relationship between large attached mast structural components (i.e., the attachment of yardarms with the main trunk). The **crossSectionAssemblies** feature, on the other hand, is used to represent main trunks, yardarms and auxiliary masts. In the **Scenegraph** tree control (see Figure 4.5 through Figure 4.7), **crossSectionAssemblies** are represented as folders, which in turn contain a series of sub-folder used to store bay, cross-section, and gusset objects. MASTSAS uses **crossSectionAssemblies** to maintain collections of **crossSections** that define the shape and orientation of the assembly component. In addition, **crossSectionAssemblies** also store information related to the bay type and configuration. **CrossSectionAssemblies** can only be created through the use of wizards. In general, these wizards prompt the user for the overall height of the component, the number of cross sections, the type and configuration of bay (lattice or enclosed), and beam/plate/stiffener properties. When creating auxiliary masts and yardarms, the wizard will also prompt the user for additional information related to the attachment and post-attachment orientation.

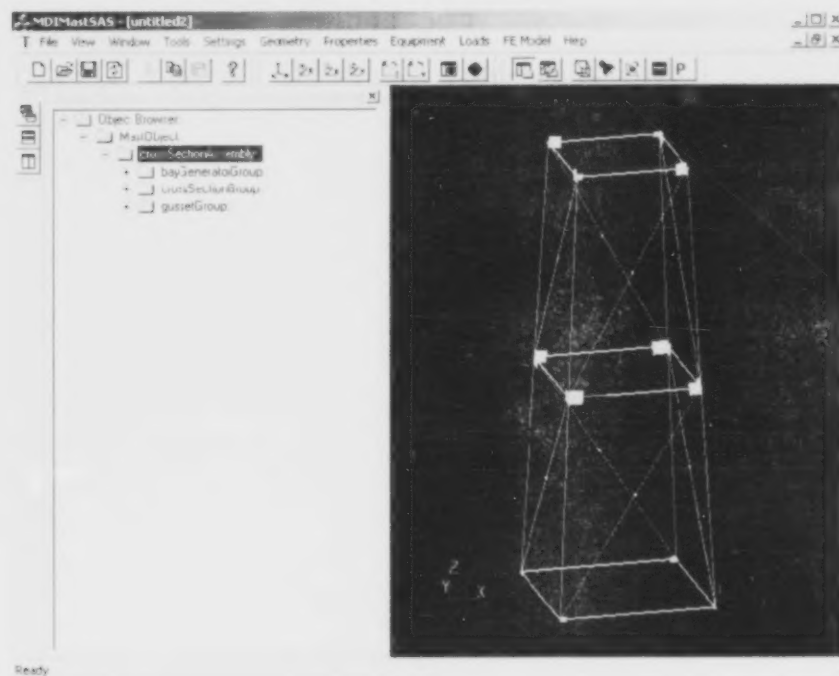


Figure 4.5: Representation of CrossSectionAssemblies in the Scenegraph Tree Control

crossSectionAssembly Properties

Object Info

Object Geometry Info

Cross Section Assembly

Data Summary

Number of Cross Sections:	5	Number of Sides:	4
Number of Bay Generators:	4	Default Bay Type:	Lattice Type A
Number of Gusset Plates:	56	Number of Attachments:	0

Cross Section Wizard

☒ Activate Cross Section Wizard
 ☐ Clear Selections
 ☐ Select Base Cross Section
 ☐ Select Top Cross Section

Elevation

Bay Generator Wizard

☐ Activate Bay Generator Wizard
 ☐ Clear Selections
 ☐ Select Base Cross Section
 ☐ Select Top Cross Section

Bay Type

Brace Display

☒ Apply to All Braces
 ☐ Apply to Braces on Edge
☒ Draw as a line
 ☐ Draw Beam Section

Brace Orientation

Rotation Angle (deg):

X Shift:

Y Shift:

☒ Horizontal Members
 ☐ Vertical Members
 ☐ Diagonal Members

Cross Section Visibility

☐ Visibility On
 ☒ Visibility Off

Gusset Plate Visibility

☐ Visibility On
 ☒ Visibility Off

End Point Visibility

☐ On
 ☐ Off
 Size:

Figure 4.7: CrossSectionAssembly Property Page (GUI)

4.4 Mast Brace End Points

As suggested by its name, the primary function of the **MastBraceEndPoint** (MBEP) is to define the start and end points for beam or brace objects. However, MBEP's are also used to define the corner points of other MASTSAS geometric objects, including stiffened panels and plates. User's can create MBEP's by selecting the "Brace End Point" option under the main menu's "Geometry" tab (see Figure 4.8) and supplying location and display size in the appropriate fields provided in the property page (see Figure 4.9).

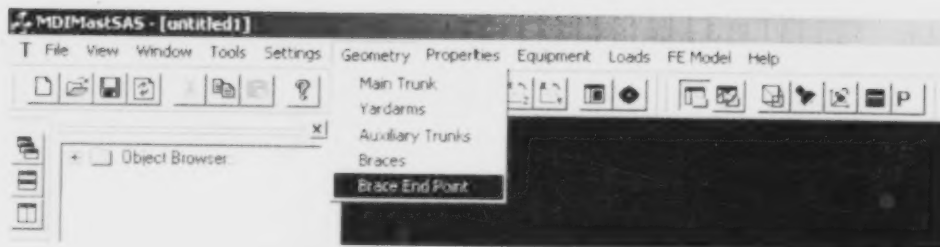


Figure 4.8: Creating a Brace Endpoint

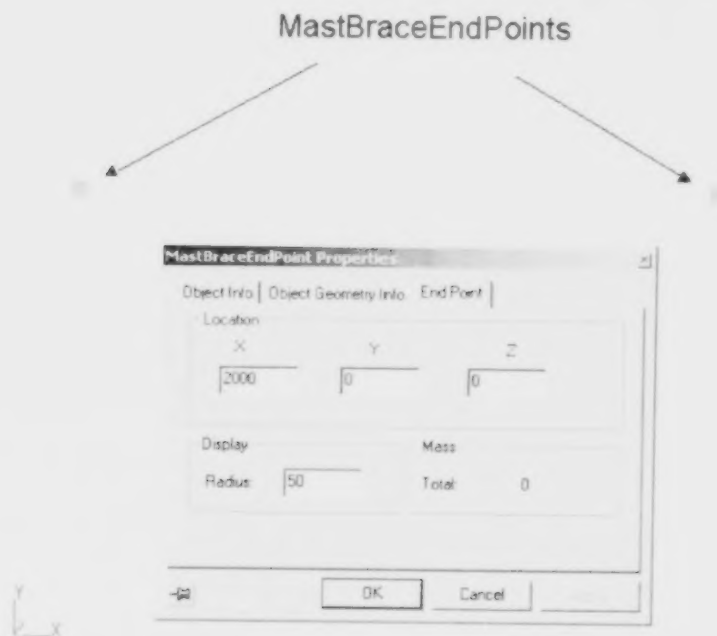


Figure 4.9: MastBraceEndPoint Property Page (GUI)

4.5 User-Generated End Points

In some modeling situations, allowing the user to directly modify the global coordinates of a **MastBraceEndPoint** may result in undesirable affects to related or attached structural components. For example, changing the coordinates of a point attached to a plate stiffener may result in the stiffener no longer being oriented in the plane of the plate. In order to prevent these and similar situations arising in MASTSAS, a **GeneratedEndPoint** was developed. The **GeneratedEndPoint** features are used in situations where it is important to restrict the way in which the coordinates of an end point are modified (keeping end points of stiffened panels in plane, for example).

4.6 Braces

Braces can be created by selecting the "Braces" option provided under the main menu's "Geometry" tab (see Figure 4.10). A wizard will then step the user through the process of selecting the two end points (see Figure 4.11), the beam section and material type (see Figure 4.12), and orientation (see Figure 4.13). It is important to note that the relationship between the MBEP assigned to the brace during its creation is maintained as long as the brace is present in the model. As a result, changes made to the geometric location of the MBEP will mean a corresponding change in the orientation of any braces attached to it.

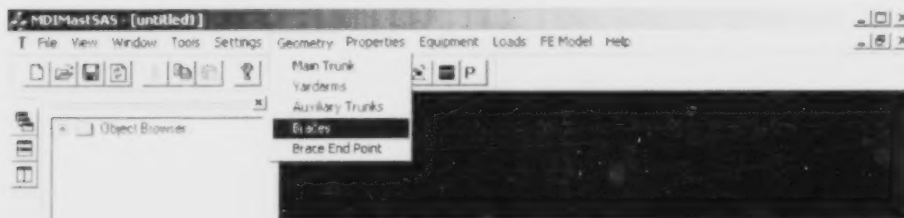


Figure 4.10: Creating a New Brace

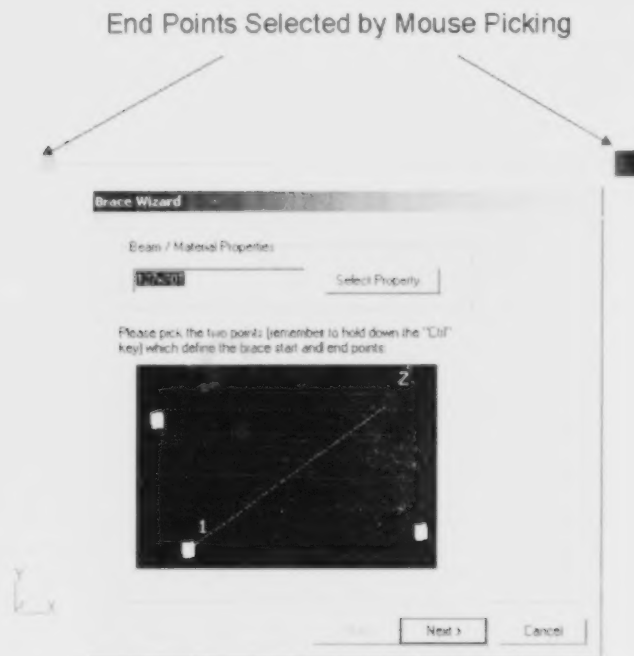


Figure 4.11: Page 1 of Brace Generator Wizard (End Point Selection)



Figure 4.12: Page 2 of Brace Generator Wizard (Brace Properties)

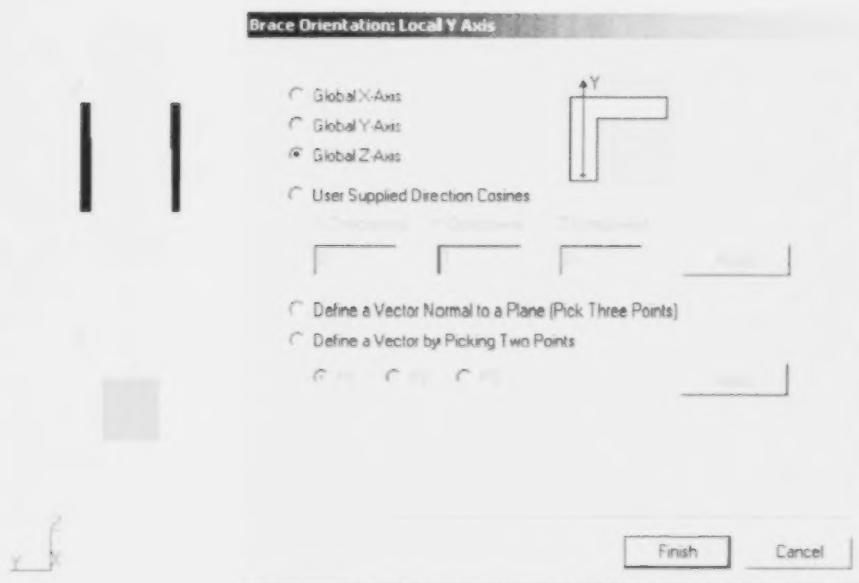


Figure 4.13: Page 3 of Brace Generator Wizard (Brace Orientation)

The GUI for a typical brace object is shown in Figure 4.14 below. Through this GUI, user's can modify the orientation and appearance of the brace. In addition, user's can also modify the section and material properties of the brace.



Brace Drawn as a Line by MASTSAS



Figure 4.14: Brace Property Page (GUI)

4.7 Bays and Bay Generators

A bay, in the context of the MASTSAS program, represents a lattice or enclosed section of a trunk, yardarm, or auxiliary mast (**crossSectionAssembly**) positioned between two cross sections (see Figure 4.15). In terms of the data hierarchy, bays are children of **crossSectionAssemblies**, and for convenience are grouped together into **BayGeneratorGroup** folders (see Figure 4.16). MASTSAS provides users with two basic forms of bay generators: lattice and enclosed. The main difference between the two types of bay generators is that lattice bay generators create a series of braces, the configuration of which depends on the particular type of lattice bay generator selected (see Figure 4.17), while enclosed bay generators create a series of four-sided stiffened panels.

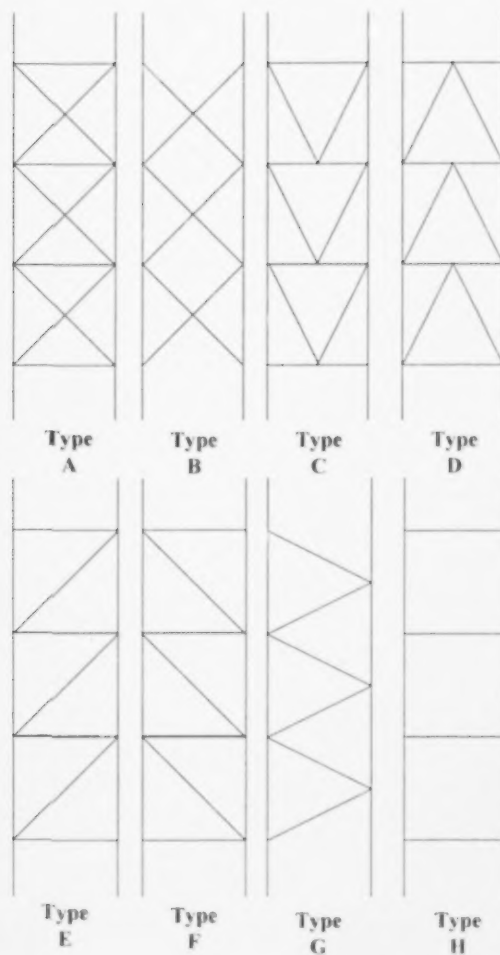


Figure 4.17: Lattice Mast Configuration Types

Once created, the graphical representation (solid or wireframe) and number of bay generator sub-divisions can be modified through its property page or GUI (see Figure 4.18 and Figure 4.19).



Figure 4.18: BayGenerator Property Page (GUI)



Figure 4.19: Doubling the Number of Bays Using the BayGenerator GUI

As mentioned above, enclosed bay generators create a series of four-sided stiffened panels. These panels form the outer walls of an enclosed mast by attaching to the sides of neighboring cross-sections used to define the shape of the trunk. In general, these four-sided stiffened panel objects are created by MASTSAS main trunk, yardarm, or auxiliary mast wizards. These wizards allow users to define the attributes of these panels with minimal effort. Once created, however, these attributes (including the material types, stiffener type, and orientation) can be modified through the panel property page or GUI (see Figure 4.20). An important feature provided in this property page is the ability to view and modify the orientation of the material axes. These material axes are used to define the local coordinate system for composite materials (see Figure 4.21).

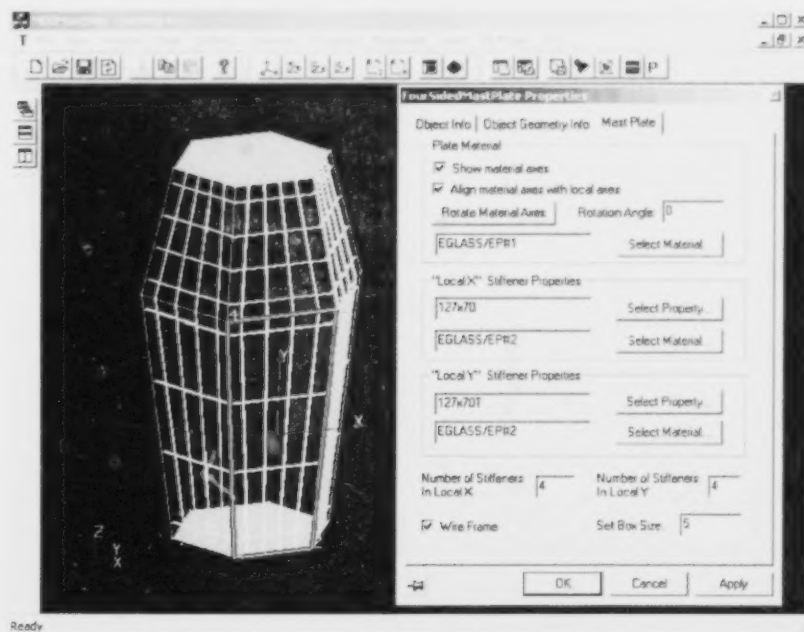


Figure 4.20: Four Sided Mast Plate (Highlighted in Purple) Property Page

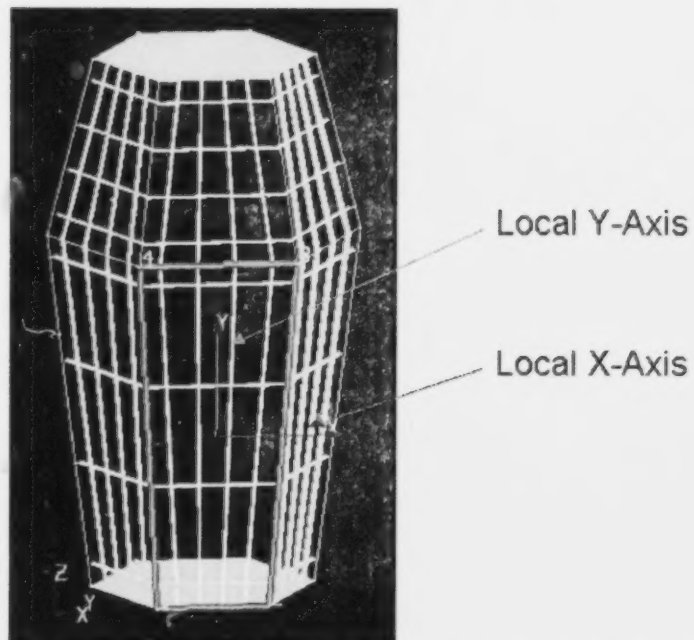


Figure 4.21: Four Sided Mast Plate Material Local Coordinate System

4.8 Gusset Plates

Gusset plates are created automatically for lattice bay types (see Figure 4.22). There are total of 8 different gusset types available in MASTSAS. The choice of gusset type depends entirely on the type of bay type selected by the user. The size of individual gusset plate is a function of the orientation, size and materials used in the relevant beams or braces. Users can override the geometry of any gusset by modifying the attribute values supplied on its' property sheet (see Figure 4.23).

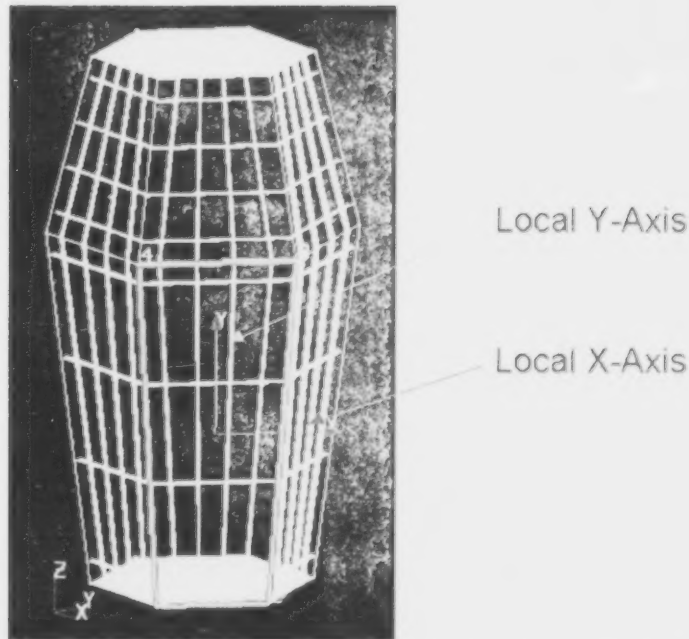


Figure 4.21: Four Sided Mast Plate Material Local Coordinate System

4.8 Gusset Plates

Gusset plates are created automatically for lattice bay types (see Figure 4.22). There are total of 8 different gusset types available in MASTSAS. The choice of gusset type depends entirely on the type of bay type selected by the user. The size of individual gusset plate is a function of the orientation, size and materials used in the relevant beams or braces. Users can override the geometry of any gusset by modifying the attribute values supplied on its' property sheet (see Figure 4.23).

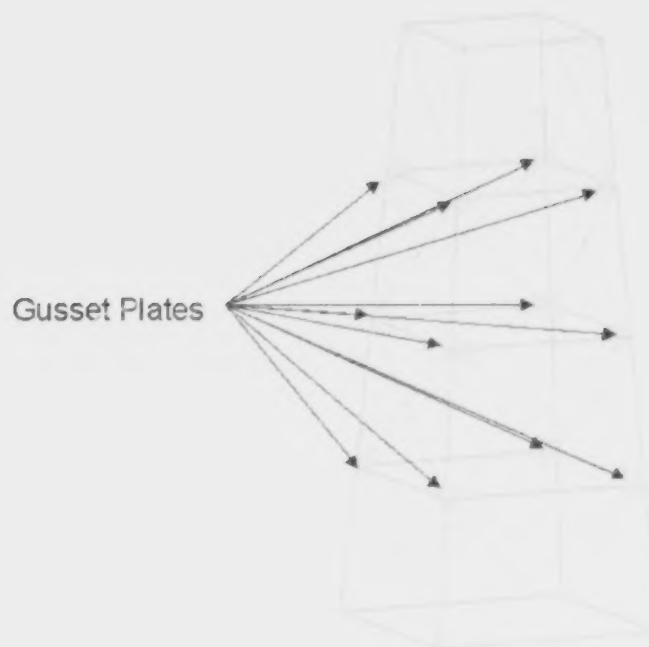


Figure 4.22: Gusset Plates Generated by MASTSAS Wizard



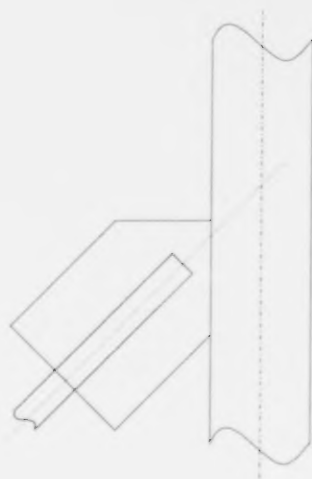
Figure 4.23: Gusset Plates Property Page

Gusset plates are used to facilitate connection of the braces/lattice members. The dimensions of the gusset plates are computed using a rule-based approach for the following types of connections (see Figure):

- GP Type 1: Brace to Column
- GP Type 2: Brace to Column base
- GP Type 3: Two-Brace to Column
- GP Type 4: Two-Brace to Beam
- GP Type 5: Brace to Beam
- GP Type 6: Brace to Beam-Column
- GP Type 7: Two-Brace to Beam-Column
- GP Type 8: Four Brace

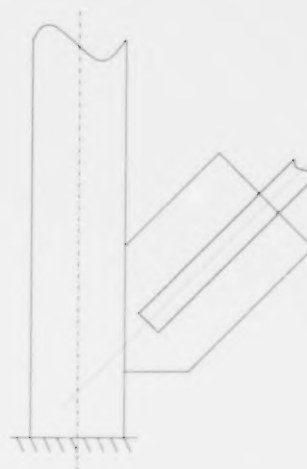
Consider, for example, the two-brace-to-beam-column connection shown in Figure (g), details of which are shown in Figure 4.25. The following rules shall be used to determine the dimensions of the gusset plate:

- (a) The orientation of the braces is such that they all meet at point O, which lies on the neutral axis of the column member. This eliminates the possibility of additional eccentricities in the column member.
- (ii) The extremities of the members (i.e. point A, edge EG and point K) shall have a clearance of at least 12.5mm (1/2") from the edge of the column.
- (iii) The lengths AB, EF, KL shall be set such that the weld capacities along these lengths are equal or greater than the ultimate strengths of the respective braces.
- (iv) The clearance denoted by ' c_2 ', extending from the extremities of the braces, shall be at least 12.5mm (1/2").
- (v) The gusset plate shall be completely defined by the coordinates of the points P, B', D', F', H', J', L' and Q; along with the plate thickness (t).



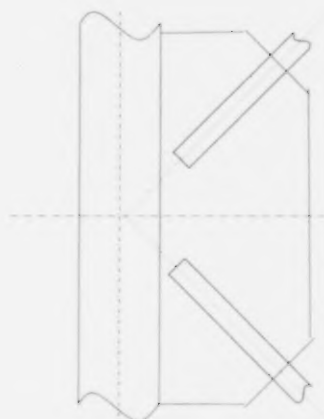
**Single Brace To Column
(GP-1)**

(a)



**Single Brace To Column Base
(GP-2)**

(b)



**Double-Brace To Column
(GP-3)**

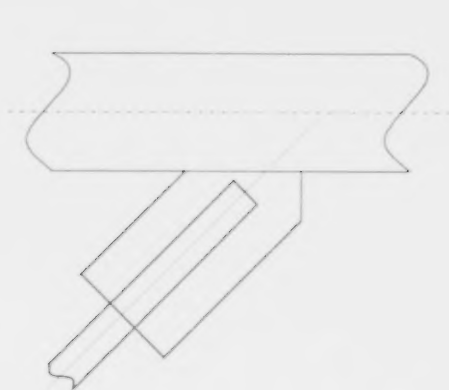
(c)



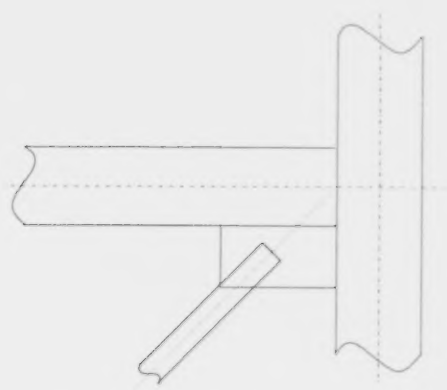
**Double-Brace To Beam
(GP-4)**

(d)

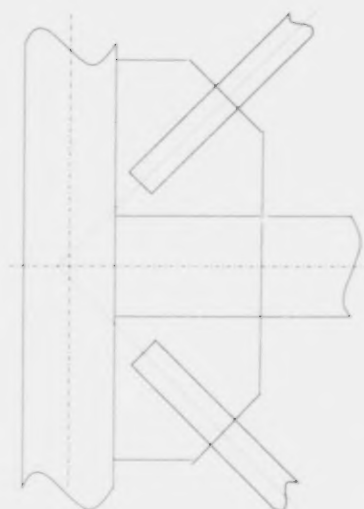
Figure 4.24: Gusset Plate Configurations: (a) GP1, (b) GP2, (c) GP3, (d) GP4



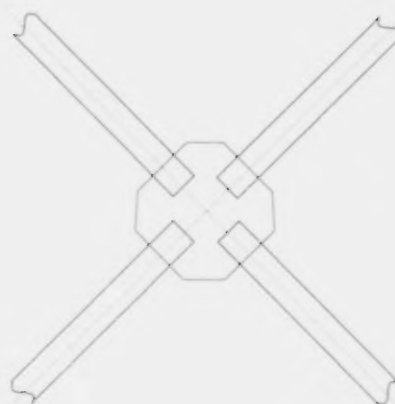
Single Brace To Beam (GP-5)
(e)



**Single Brace To Beam-Column
(GP-6)**
(f)



**Double-Brace To Beam-
Column (GP-7)**
(g)



Four Brace (GP-8)
(h)

Figure 4.24: Gusset Plate Configurations (cont.): (e) GP5, (f) GP6, (g) GP7, (h) GP8

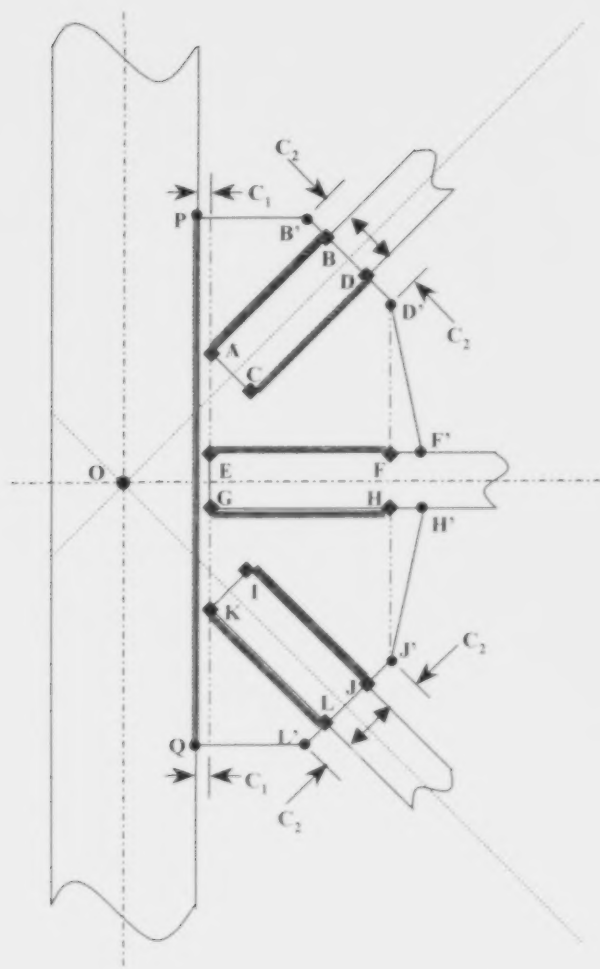


Figure 4.25: Details of Gusset Plate (Type GP7)

4.9 Multisided Stiffened Panels and Cutouts

In addition to the four-sided stiffened panels, users can also define multi-sided stiffened panels, denoted hereafter as MSSP's. Existing cross-sections are used to provide the overall shape for the MSSP's. The process of creating MSSP's starts with the selection of the cross-section upon which the MSSP will be attached. Once the property page for the desired cross-section is visible (see Figure 4.26) the MSSP can be created by selecting the "Add Stiffened Panel" button. At this point the MASTSAS model will be re-drawn to include the MSSP (see Figure 4.27 and Figure 4.28) and the property page for the stiffened panel will then be presented to the user (see Figure 4.29). Users can then

modify the attributes of the MSSP, including stiffener orientation, material properties, and display attributes. It is important to note that only one MSSP can be assigned to any single cross-section.

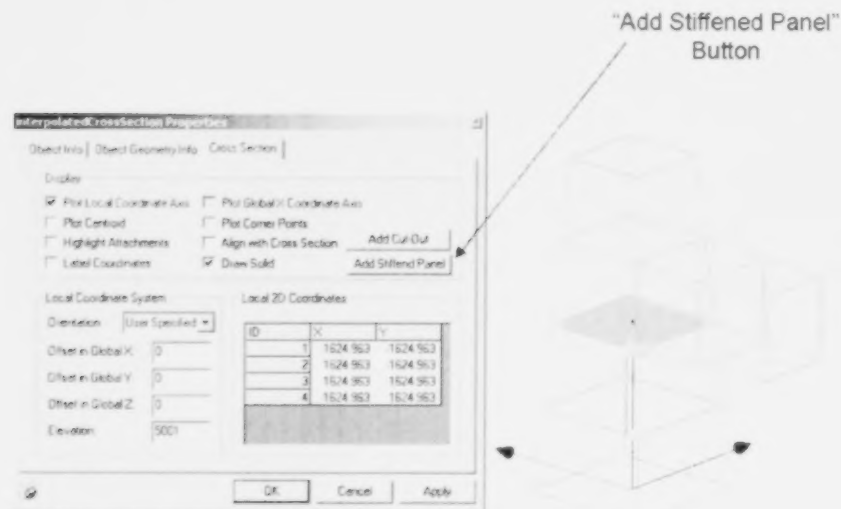


Figure 4.26: Adding Stiffened Panels to Cross-Sections

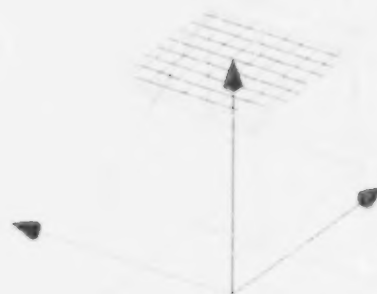


Figure 4.27: Mast Model Including New Stiffened Panel

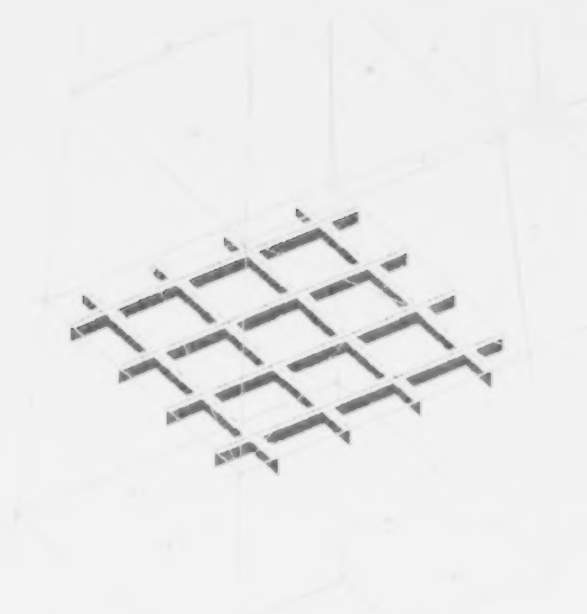


Figure 4.28: Close-Up of Stiffened Panel

Multi-Sided Stiffened Panel Properties

Object Info | Object Geometry Info | Stiffened Panel

Brace Spacing and Orientation:

Local X: Local Y:

Angle:

Stiffeners Local X and Y Dimensions:

Local X: Local Y:

"Local X" Stiffener Properties:

Section:

Material:

"Local Y" Stiffener Properties:

Section:

Material:

Plate Properties:

Material:

Thickness:

Display:

☐ Display Node Numbers ☐ Display Local Origin

☐ Display Beam Sections ☐ Display Local Axis

Figure 4.29: Multi-Sided Stiffened Panel Property Page

Once an MSSP has been created, one or more cutouts can be assigned. Like MSSP's, cut-outs are the 'children' of cross-sections. However, unlike MSSP's, individual cross-sections can have more than a single cutout. The process of creating cutouts is very similar to that used to create MSSP's. Users must first select the appropriate cross-section, display that cross-section's property page, and then click the "Add Cut-Out" button. MASTSAS will then create a new cutout and display the GUI for this cutout to the user (see Figure 4.30 and Figure 4.31). Users can then define the location and radii of the cutout corners (in terms of local coordinates).

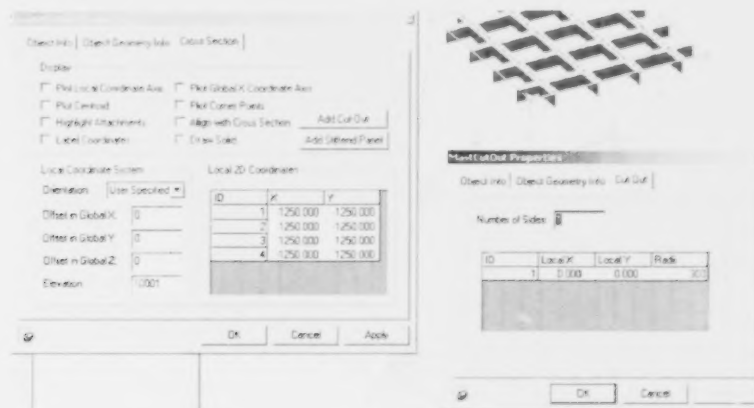


Figure 4.30: Defining Cutouts



Figure 4.31: MASTSAS-Generated FE Meshes with Cutouts

Users should note that because the geometry of both MSSP's and cutouts are defined relative to a 'parent' cross-section, modifications made to the cross-section may result in unexpected MSSP's and cutout behavior.

4.10 Mast Geometry Modelling Wizards

In order to simplify the generation of complex structural components, MASTSAS provides users with a number of geometric modeling wizards, which can create and assemble complex structural components with limited interaction with the user. A brief description of each wizard is provided in the section below

4.10.1 Cross Section Assembly Wizard

CrossSectionAssembly wizards are used to rapidly generate large structural components such as main trunks, yardarms, and auxiliary trunks. Users can execute these wizards by selecting one of the options provided under the geometry tab of the MASTSAS main menu (see Figure 4.32). Initially, all three wizards prompt the user for the general shape (number of sides and cross-sections) and dimensions (overall height, length of sides, etc), as depicted in Figure 4.33. Following this, the wizard prompts the user for the type (defined or interpolated) and dimensions of each cross-section (see Figure 4.34). Users are then prompted for the type of bay generators to be used (see Figure 4.35). It is important to note that MASTSAS allows users to define **crossSectionAssemblies** can have more than one type of bay generator (see Figure 4.36 and Figure 4.37). For example, a main trunk having three cross-sections could have lattice configuration 'A' as the bay type between cross-sections 1 and 2, while having an enclosed bay generator between cross-sections 2 and 3 (see Figure 4.38)

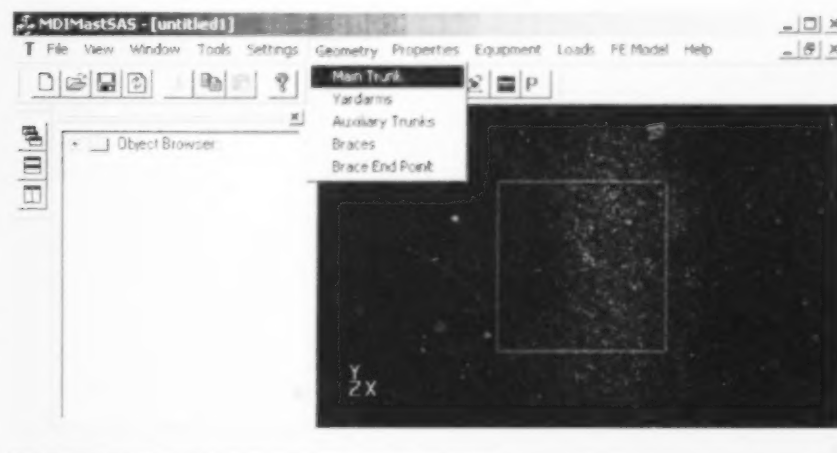


Figure 4.32: Creating a New Main Trunk (CrossSectionAssembly)

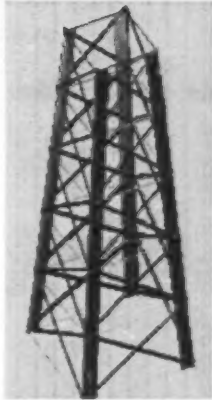
Trunk / Yardarm / Auxiliary Mast Wizard

Main Trunk Name:
Main Trunk

Number of Sides:
4

Overall Height:
10000

Number of Cross Sections:
10



< Back Next > Cancel

Figure 4.33: Defining Overall Dimensions of the Main Trunk

Cross Section Properties

Cross Section 1 of 5

☒ Define cross section by specifying length of sides

☐ Enter cross section points manually

Enter "Lx" or Height of Cross Section
(where Lx represents the Longitudinal axis of the Trunk, Yardarm, or Auxiliary Trunk)

0



Enter Dimension Lx: CSU

Enter Dimension Ly: CSU

< Back Next > Cancel

Cross Section Properties

Cross Section 3 of 5

☒ Define cross section by specifying length of sides

☒ Enter cross section points manually

☐ Enter cross section points manually

Enter "Lx" or Height of Cross Section
(where Lx represents the Longitudinal axis of the Trunk, Yardarm, or Auxiliary Trunk)

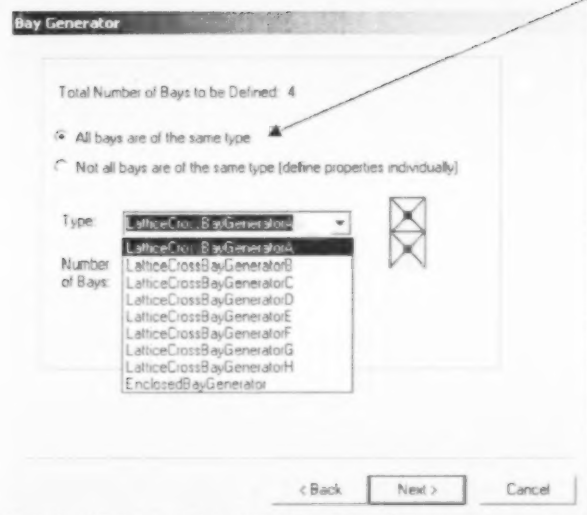
5000

< Back Next > Cancel

Defined Cross-section

Interpolated Cross-section

Figure 4.34: Defining Cross-Sectional Properties



Selecting this Option will ensure all Bay Generators will be of the same type

Figure 4.35: Defining the Bay Generators

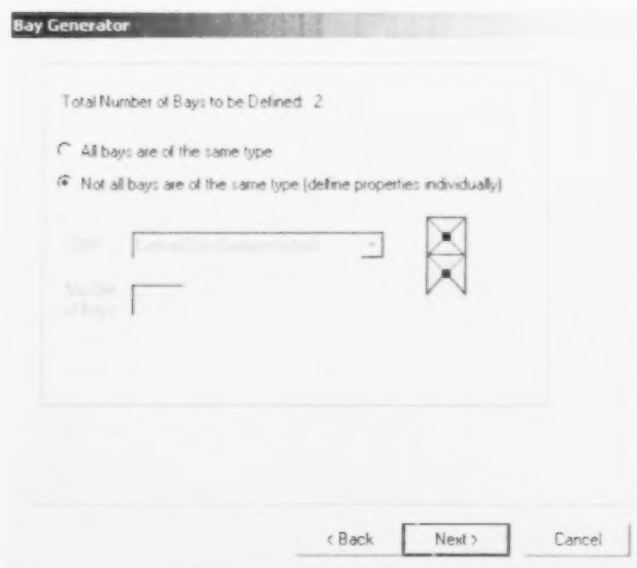


Figure 4.36: Generating a CrossSectionAssembly with Multiple Bay Generator Types

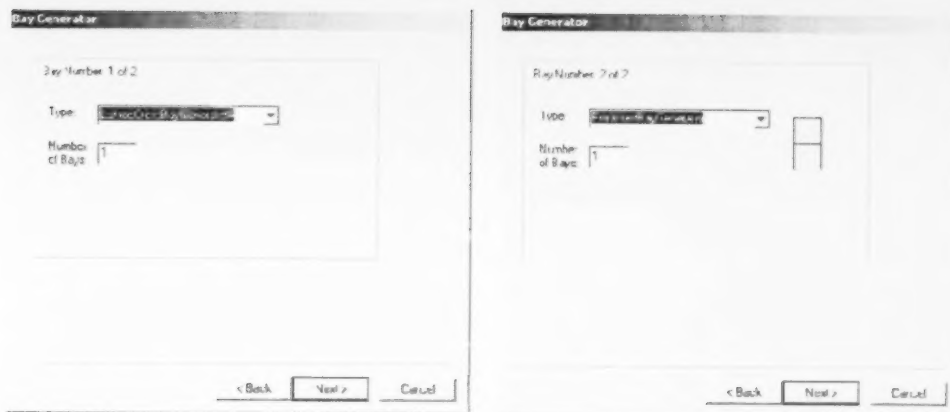


Figure 4.37: Screen Captures of CrossSectionAssembly Wizard Dialog Boxes Used to Define Multiple Bay Generator Types

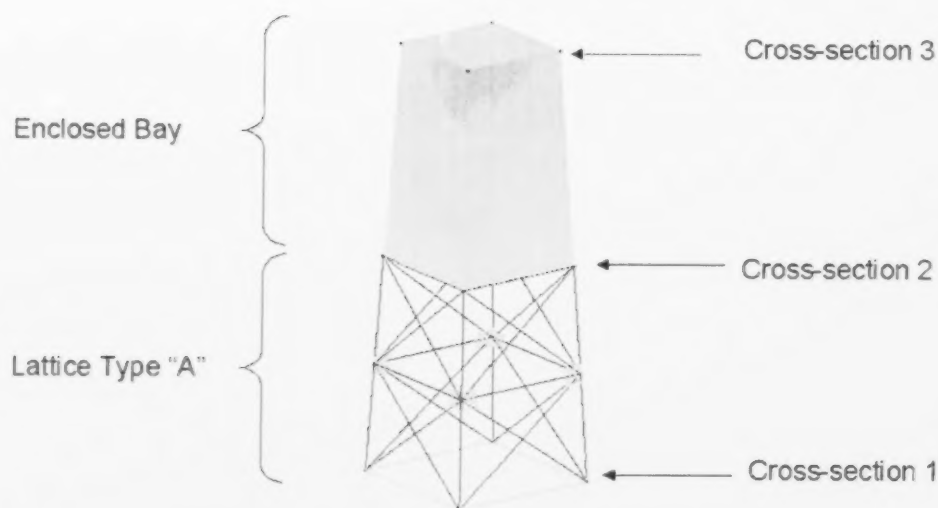


Figure 4.38: Main Trunk with Two Different Bay Types

Following the selection of the bay types, users next supply information related to the brace material and section types (see Figure 4.39), or in the case where enclosed bay generators have been defined, plate and stiffener data (see Figure 4.40 and Figure 4.41).

Material/Property Definition

Legs

Steel1	Select Material...
127x70T	Select Property...

Diagonal Members

Steel1	Select Material...
CirSect:Ro=125:Ri=75	Select Property...

Horizontal Members

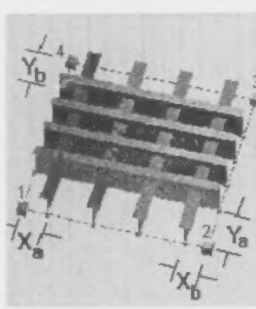
Steel1	Select Material...
127x70T	Select Property...

< Back Next > Cancel

Figure 4.39: Assigning Lattice Bay Generator Properties Using the Wizard

Enclosed Bay Stiffener Orientation

Number of Stiffeners in Local X direction	4
Number of Stiffeners in Local Y direction	5
length "Xa"	200
length "Xb"	200
length "Ya"	300
length "Yb"	300



< Back Next > Cancel

Figure 4.40: Assigning Enclosed Bay Generator Stiffener Orientation Using the Wizard

Enclosed Bay Stiffener Properties

Plate Properties

EGLASS/EP#1 Select Plate Material ...

Plate Thickness: 25.4 (Not used for Laminates)

"Local X" Stiffener Properties

EGLASS/EP#1 Select Material ...

127x70I Select Beam Section ...

"Local Y" Stiffener Properties

EGLASS/EP#2 Select Material ...

127x70I Select Beam Section ...

< Back Next > Cancel

Figure 4.41: Assigning Enclosed Bay Generator Stiffener Properties Using the Wizard

Once all the above information has been supplied, the wizard then prompts the user for the orientation of the newly created **CrossSectionAssembly**. For cases involving the modeling of the main trunk, the only option made available to the user is to align the local "z" or vertical axes parallel to the global "Z" axis (see Figure 4.42). In cases involving yardarms or auxiliary trunks, however, the user must 'attach' these components to existing structure. For these cases the wizard prompts the user for both the post-attachment orientation of the yardarm or auxiliary trunk (port, aft, etc.), as shown in Figure 4.43, and the location of the attachment (defined by two existing cross sections. Users are asked to select the attachments by 'picking' upper and lower cross-sections on the main trunk or other yardarms or auxiliary masts (see Figure 4.44 and Figure 4.45).

Orientation

Method for Defining Orientation

☐ Orientation Defined in Terms of Ship's Head (Not Recommended)
☒ Orientation Defined in Terms of Global Coordinate System

☒ Vertical (Default Orientation for Main/Aux Trunks)
☐ Forward (Bearing Angle = 0 Degrees)
☐ Starboard (Bearing Angle = 90 Degrees)
☐ Aft (Bearing Angle = 180 Degrees)
☐ Port (Bearing Angle = 270 Degrees)
☐ User Defined Bearing Angle:

☐ Apply Azimuth: Azimuth:

☐ Add Roof

Create Cancel

Figure 4.42: Assigning Main Trunk Orientation Using the Wizard

Orientation

Method for Defining Orientation

☐ Orientation Defined in Terms of Ship's Head (Not Recommended)
☒ Orientation Defined in Terms of Global Coordinate System

☐ Forward (Bearing Angle = 0 Degrees)
☒ Starboard (Bearing Angle = 90 Degrees)
☐ Aft (Bearing Angle = 180 Degrees)
☐ Port (Bearing Angle = 270 Degrees)
☐ User Defined Bearing Angle:

☐ Apply Azimuth: Azimuth:

☐ Add Roof

Forward (0°)
 Bearing Angle
 Port (270°) Stbd (90°)
 Aft (180°)

Figure 4.43: Assigning Yardarm Orientation Using the Wizard

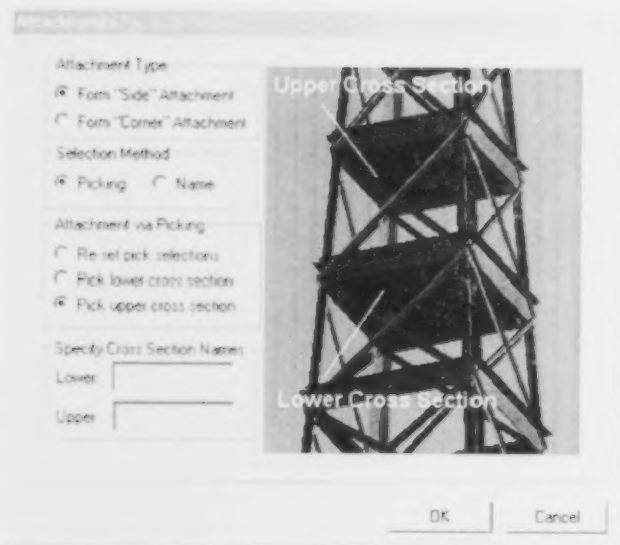


Figure 4.44: Assigning Yardarm Attachment Location Using the Wizard



Figure 4.45: Resulting Yardarm After Attachment with Main Trunk

4.10.2 Brace Wizard

While, in general, most braces are created automatically by the **CrossSectionAssembly** wizards, users also have the option of creating individual braces using the MASTSAS brace wizard. Creating braces using the wizard involves a three-step process. First, users select the two end points of the brace by picking existing **MastBraceEndPoints** (recall Figure 4.11). Next, the user is prompted for the material property to be assigned to the new brace (recall Figure 4.12). Note that the default type is used to define the section property. In the final step, users are then asked to define the local orientation of the brace (recall Figure 4.13). Once created, users can then modify the attributes of the new brace (including the section type) through its property page (recall Figure 4.14).

4.10.3 Brace End Point Wizard

Creating new **MastBraceEndPoints** using the MASTSAS supplied wizard is very simple. The wizard only requires users to supply the global coordinate of the end point (recall Figure 4.9).

5. EQUIPMENT

A database of equipment suitable for attachment is provided in the "equipmentDB.edm" file (part of the MASTSAS installation material/files). Users can view the properties of the equipment provided in this file by selecting the "View Database" option located under the "Equipment" tab in the main menu (see Figure 5.1). Attaching the various pieces of equipment defined in the database to a MASTSAS geometric model is controlled by the **MastEquipmentCollector**. The property page for the **MastEquipmentCollector** also provides the user with the list of currently attached equipment, along with tools for modifying and deleting these objects (see Figure 5.2).

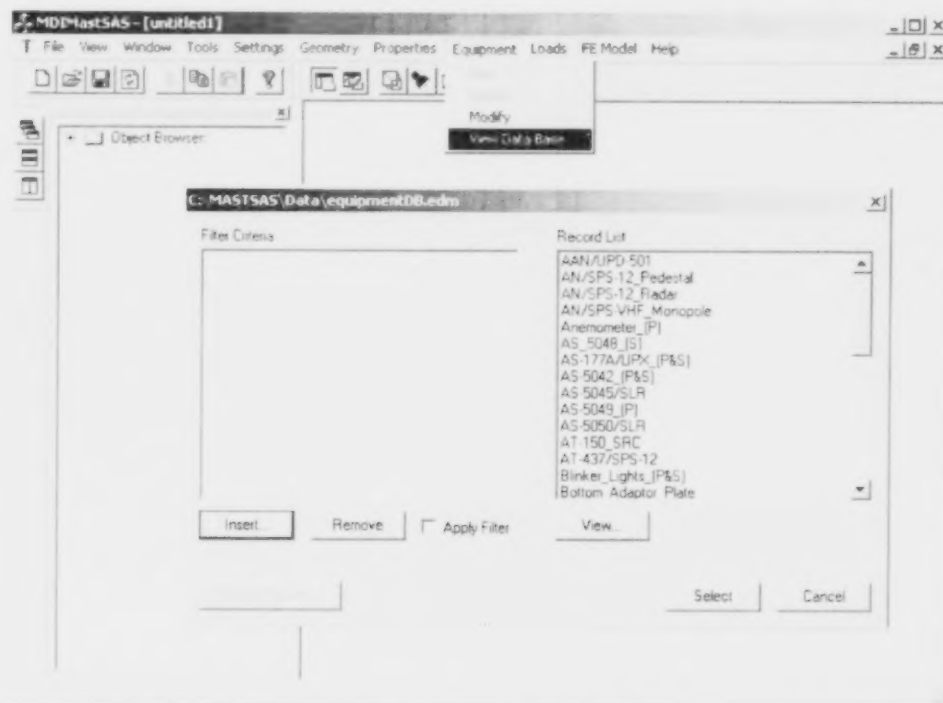


Figure 5.1: Viewing the Contents of the Equipment Database



Figure 5.4: Defining the Equipment Attachment Location and Orientation

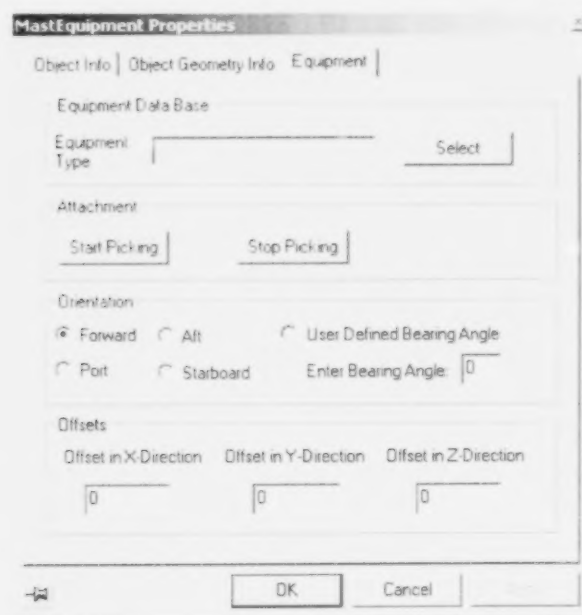


Figure 5.5: MastEquipmentObject Property Page

6. BOUNDARY CONDITIONS

The management of boundary condition data is the responsibility of the MASTSAS **BoundaryConditionCaseManager**. The **BoundaryConditionCaseManager** provides user's with a tool for creating, modifying, and deleting boundary constraints applied to end points. In order to view the current status of the **BoundaryConditionCaseManager**, users can display its' property page (see Figure 6.1) by first selecting the "BC's" tab on the MASTSAS main menu.



Figure 6.1: BoundaryConditionCaseManager Property Page

The three most important options provided by the **BoundaryConditionCaseManager** property page are (i) the ability to create sets of different boundary constraint scenarios (i.e., clamped base or pinned base) and store them in the model database, (ii) delete a an existing set of boundary constraint data, and (iii) assign one of the boundary constraint cases as the active case. The significance of making one of the boundary constraint cases active is two-fold. First, only the active constraints are plotted (the red lines of Figure 6.2 represent fixed degrees of freedom, while the white lines of Figure 6.3 represent pinned degrees of freedom), and secondly, only the active constraints are applied to the finite element model.



Figure 6.2: Model Showing Fixed Boundary Condition (Red Lines)

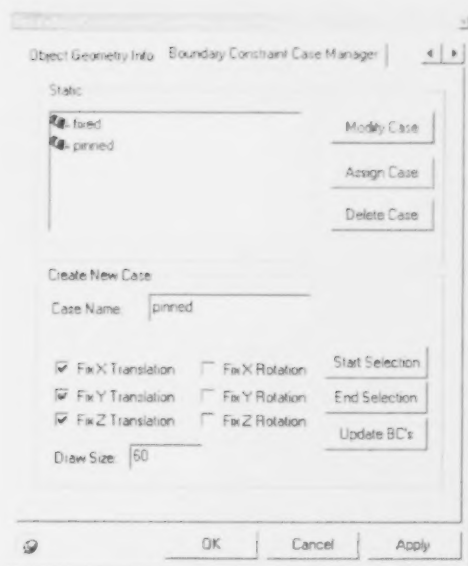


Figure 6.3: Model Showing Pinned Boundary Condition (White Lines)

7. LOADS

7.1 Load Case Manager

MASTSAS provides users with wizards for creating a variety of different loading conditions, including: self weight, loads due to ship motion, wind loads, blast loads, underwater shock loads, and live loads. These wizards can be accessed by clicking the "New Case" button located on the **LoadCaseManager** property page (see Figure 7.1), which can be displayed by selecting the "Modify" option listed under the main menu "Loads" tab.



Figure 7.1: LoadCaseManager Property Page

Initially, the load case wizard provides the user with a list of the six different types of MASTSAS loads (see Figure 7.2). Once the user has identified the desired load type (wind loading, for example), the wizard prompts the user for data specific to that load type (see Figure 7.3 through Figure 7.8). Once the user has entered all the relevant data, the load case wizard returns control back to the **LoadCaseManager**.

6 Different Load
Case Types

Select Load Case Types

Load Case Name: **Self Weight (Static Point)**

- ☒ Self Weight
- ☐ Ship Motion
- ☐ Wind Load
- ☐ Blast
- ☐ Underwater Shock
- ☐ Live Static Point Loads

Next Cancel

Figure 7.2: Initial Page of the LoadCase Wizard

Ship Motion Data

Definition Options

- ☒ User Defined
- ☐ Based on Design Loads (Data provided in Load Database)
- ☐ DMEM 10

Acceleration Components

Surge: Sway: Heave:

Roll: Pitch: Yaw:

Center of Rotation

Speed: LBP: Beam:

Xc: Yc: Zc:

Finish Cancel

Figure 7.3: Ship Motion Loads Wizard Page

Wind Load Data

Wind Speed: Knots

Air Density: kg/min3

Wind Direction

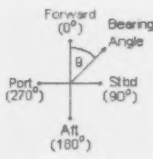
☒ Alt

☐ Starboard

☐ Forward

☐ Port

☐ Other: Enter Bearing Angle



Forward (0°) Bearing Angle

Starboard (90°)

Aft (180°)

Port (270°)

Finish Cancel

Figure 7.4: Wind Loads Wizard Page

Air Blast Data

Peak Over Pressure

☐ Peak Over Pressure and Times Known ☒ Charge and Stand-Off Distance Known

Peak Over Pressure:

Over Pressure Duration:

Dynamic Pressure Duration:

Peak Dynamic Pressure:

Charge Type:

Charge Size: (MT)

Stand-Off Distance: (km)

Show Charts

Blast Parameters

Friedlander Decay:

Atmospheric Pressure:

Temperature:

Blast Location

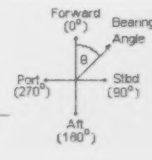
☒ Alt

☐ Starboard

☐ Forward

☐ Port

☐ Other (Bearing Angle):



Forward (0°) Bearing Angle

Starboard (90°)

Aft (180°)

Port (270°)

Blast Parameters

Shock Vel: Reflected Press: Peak Impact Pressure:

Reflected Vel: Impact Press: 1 MT StandOff:

Finish Cancel

Figure 7.5: Air Blast Loads Wizard Page

Underwater Shock Data

Definition Options

☐ Pressure Components of a point load specified
☐ Shock Spectrum Method (SSM)
☒ Base Acceleration Method (BAM)

BAM Acceleration Components

Sway: mm/s² Pitch: 1/s²
 Heave: mm/s² Yaw: 1/s²
 Surge: mm/s² Roll: 1/s²

Center of Rotation

Longitudinal: Vertical:

Finish Cancel

Figure 7.6: Underwater Shock Loads Wizard Page

Live Load Attachment Point

Live Load Attachment Point

☒ Apply Point Loads Start Picking
☐ Apply Surface Loads Stop Picking Update Data

Point Loads

	X Direction	Y Direction	Z Direction
Force	0	0	-1000
Moment	0	0	0

Pressure Loads

Node 1	Node 2	Node 3	Node 4

Finish Cancel

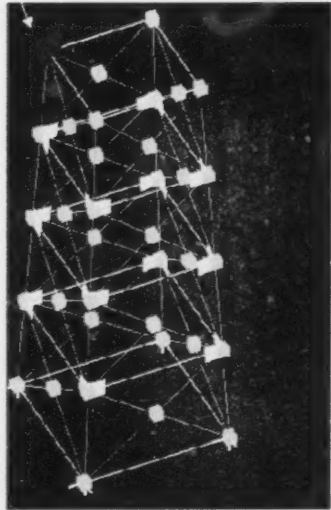


Figure 7.7: Live Loads Wizard Page

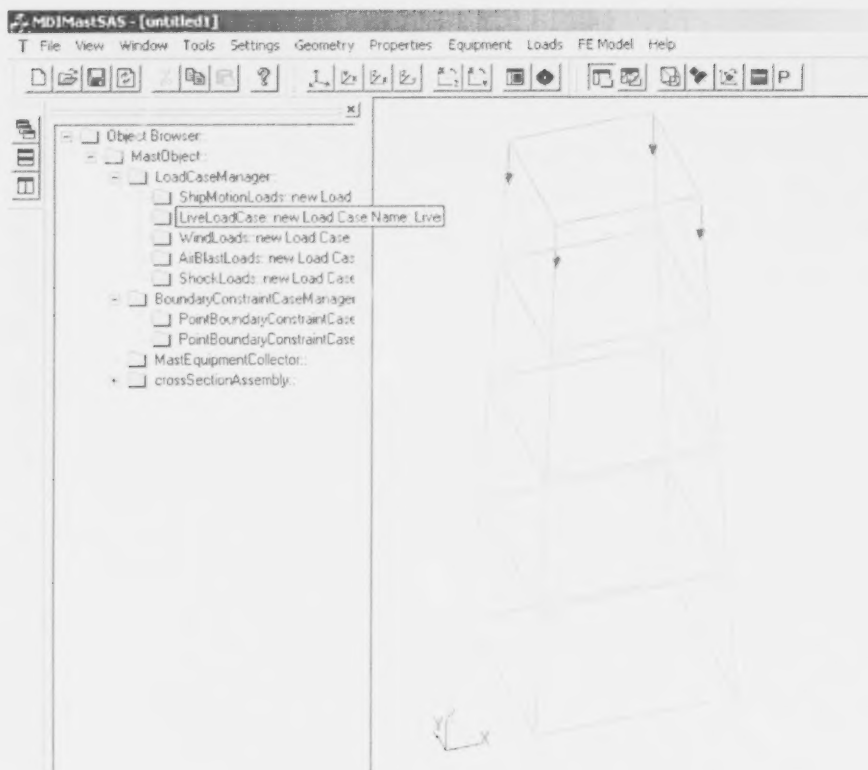


Figure 7.8: MASTSAS Display of Live Point Loads

In the case of wind loads applied to lattice masts (see Figure 7.9), MASTSAS provides a limited post-processing capability in terms of plotting critical wind velocities and vortex shedding frequencies.

The **LoadCaseManager** has been designed to act in a manner similar to the **BoundaryConstraintCaseManager** described above. Moreover, like the **BoundaryConstraintCaseManager**, the **LoadCaseManager** allows users to review the current set of cases that reside in the database, delete load cases, and even assign an active load case for use in finite element analysis.

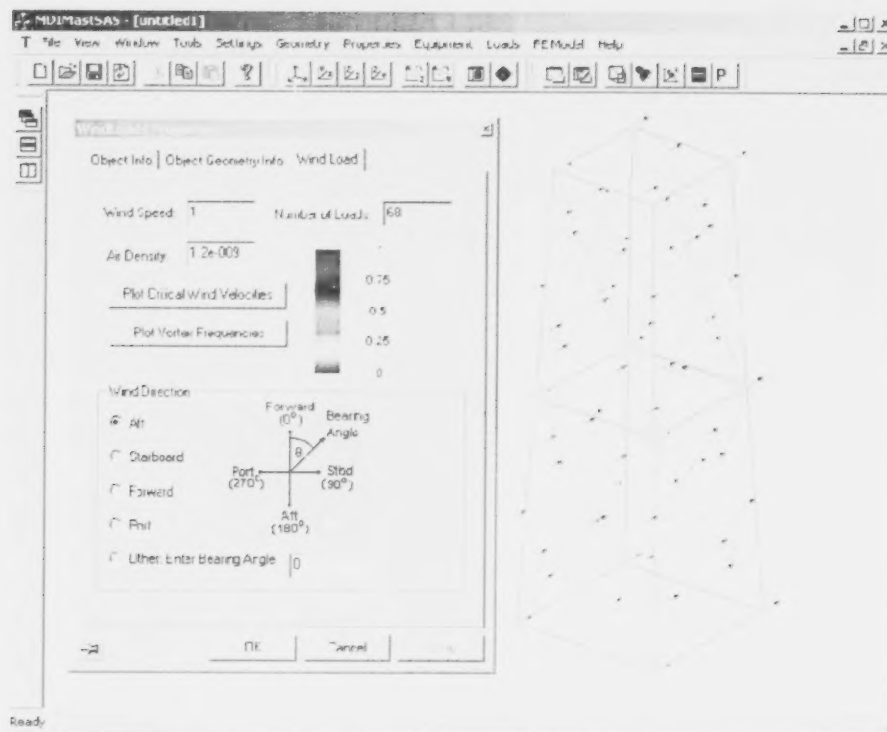


Figure 7.9: Wind Load Post Processing Capabilities

7.2 Self Weight and Live Loads

Self-weight, if required, is computed automatically using the geometry and material properties specified. No additional information is required to define the self-weight of the structure. Live loads on the mast structure may include loads due to snow, personnel, or other sources. These loads are treated as concentrated or distributed loads applied directly to the mast geometry. Concentrated loads are defined in terms of six force and moment components in the global directions (F_x , F_y , F_z , M_x , M_y , M_z) and may be applied at predefined points on the mast geometry.

7.3 Loads Due to Sea Generated Ship Motion

A ship in rough water exhibits rigid body motions as it responds to waves. The response of the ship depends on the ship's speed, dimensions, form, mass, and center of gravity location, as well as the form, height, regularity, and frequency of the ocean waves (Design Standard, DMEM 10) to which it is subjected.

It is assumed that design loads (due to sea generated ship motion) for various classes of ships are available, and are provided in a Mast Load Database. The loads are presented

in the form of six acceleration components ($a_x, a_y, a_z, \theta_x, \theta_y, \theta_z$), representing the rigid body motions of sway, heave, surge, pitch, yaw, and roll, respectively.

The MASTSAS user may choose an appropriate load from the database as a load case, but also has the option to define the loads directly. In this case, the user either provides the six acceleration components ($a_x, a_y, a_z, \theta_x, \theta_y, \theta_z$) explicitly, or provides data that can be used to compute the acceleration components using simplified equations suggested in DMEM 10. The data to be provided for this case include:

- Vertical acceleration amidship (obtained based on the ship geometry and speed, using the chart in Figure 7.10); and
- x-, y-, and z- coordinates of the mast from the centreline amidship position at the design load waterline, where x is measured positive forward and z is measured positive upwards.

Then, the x-, y-, and z-direction acceleration components are given by

$$a_x = \pm a_0 \sqrt{0.25 + 2.5(y/L)^2 + (0.7 - L/1200 + 3z/L)^2}$$

$$a_y = \pm a_0 \sqrt{0.25 + 2.5(x/L + 0.1)^2 + (1 + 0.6z/B)^2}$$

$$a_z = \pm 0.95 a_0 \sqrt{1 + 10(x/L + 0.1)^2 + (1 + 0.6z/B)^2}$$

where L and B represent the length and width of the ship, respectively.

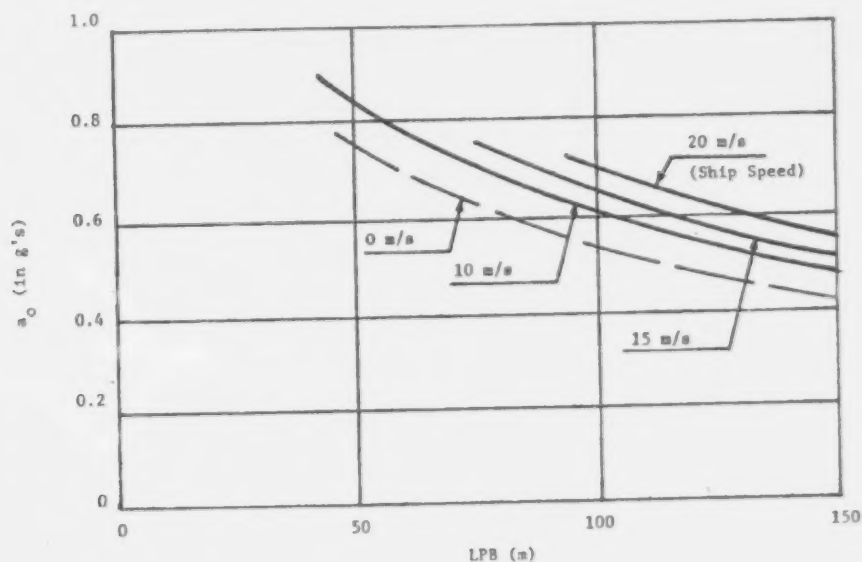


Figure 7.10: Vertical Accelerations Amidships (DMEM 10)

7.4 Wind Loads

The parameters required to define the mast wind loading consist of:

- Wind speed;
- Wind direction (aft, starboard, quarter starboard, etc.);
- Drag coefficient for various components; and
- Air density.

The approach for computing the wind load is in accordance with the methodology used in the previous Martec study by Crocker et al (1998). The wind-generated load applied to the mast components is based on the corresponding drag loading, which is computed as

$$F_D = 0.5 C_D \rho V^2 A$$

where F_D is the drag force;
 C_D is the drag coefficient;
 ρ is the air density;
 V is the wind velocity; and
 A is the projected cross sectional area along the member length

Default values of drag coefficient for various member shapes are provided as follows:

- For circular members: $C_D = 1.20$
- For non-circular members: $C_D = 2.00$
- For gusset plates: $C_D = 1.25$

The default value for the density of air is taken as 1.205 kg/m^3 ($1.205 \times 10^{-12} \text{ N s}^2/\text{mm}^4$) at 20°C . However, the user also has the option to modify any of these values, if required. As was stated previously, the load is applied to the geometry. The finite element load data is then defined, using the HOOD system, as part of the finite element model generation for the specified finite element solver being utilized for the analysis.

7.5 Air Blast Loads

The parameters required to define the air blast loads include:

- Peak overpressure;
- Duration of the positive overpressure phase;
- Stand-off distance;
- Blast location (aft, starboard, quarter starboard, etc.);
- Friedlander decay constant;
- Temperature and pressure
- Drag coefficient for various components; and
- Air density.

7.5.1 Blast Speed and Over Pressure

The approach for computing the blast parameters is in accordance with the provisions of the design standard (Norwood, 1977) employed for lattice ship masts. The design criteria for specific warships is typically specified in terms of the peak overpressure (p_0) corresponding to a 1-MT nuclear air burst at a temperature of 60°F (15.5°C), at atmospheric pressure (p_A) of 14.7 psi (101 kPa), and a stand-off distance that is chosen from Figure 7.11. The velocity of the shock front is computed using the following relation

$$U = U_s \left[1 + \frac{6p_0}{7p_A} \right]^{1/2}$$

where, p_A represents atmospheric pressure in kPa, and U_s denotes the speed of sound in air (in m/s) at the specified ambient temperature T (in °C), which is in turn given by the relation $U_s = 332 + 0.6T$ (in m/s). Following passage of the shock front, the free-field overpressure is described by the expression:

$$p(t) = p_0 \left[1 - \frac{t}{t_d} \right] e^{-kt/t_d}$$

where t_d is the duration of the overpressure positive phase, and k represents the Friedlander decay constant, which varies between 1.0 and 2.0.

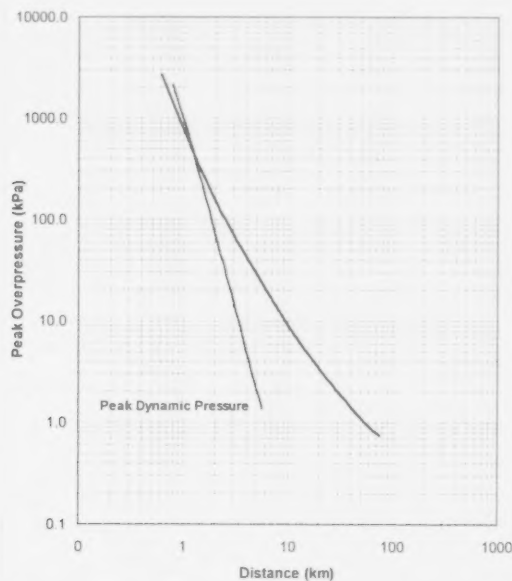


Figure 7.11: Overpressure versus Range for 1-MT Weapon (Adapted from Biggs (1964))

7.5.2 Dynamic Pressure

The passage of the shock front is accompanied by a high-velocity air stream directed away from the blast source, giving rise to a dynamic pressure. The time history of this dynamic pressure may be described by

$$q(t) = q_0 \left[1 - \frac{t}{t_{dd}} \right]^2 e^{-2kt/t_{dd}}$$

Where t_{dd} is the dynamic pressure positive phase duration, obtained using Figure 7.12 at the appropriate stand-off (i.e., range), and q_0 is the peak dynamic pressure given by

$$q_0 = \frac{5}{2} \left[\frac{p_0^2}{p_0 + 7p_A} \right]$$

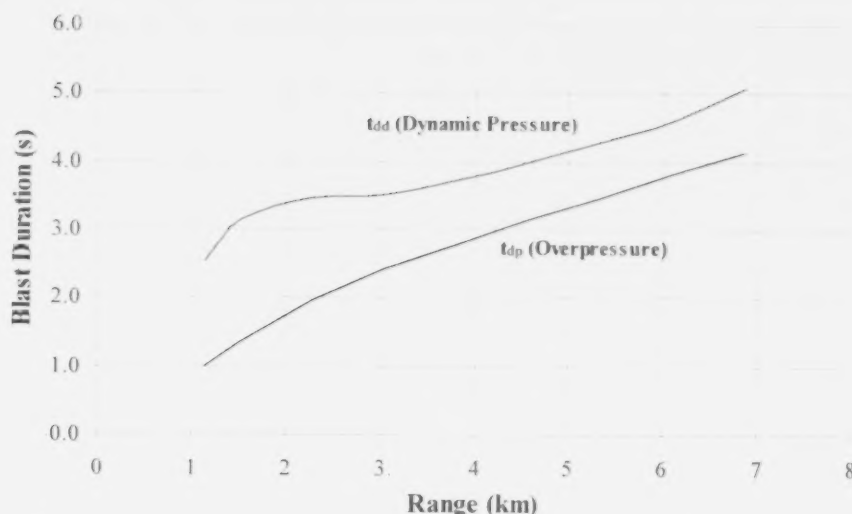


Figure 7.12: Overpressure and Dynamic Pressure Positive Durations Versus Range (1 MT Weapon) (Biggs 1964)

7.5.3 Diffraction Phase Pressure Loading

When a shock front strikes a solid surface normal to the direction of shock travel, there is an instantaneous rise in pressure above that of the shock front itself. The total pressure, referred to as the reflected pressure, is given by

$$p_r = 2p_0 \left[\frac{7p_A + 4p_0}{7p_A + p_0} \right]$$

For lattice mast structures, it is recommended that the diffraction phase pressure be incorporated as an initial impulse, for simplicity. For plates such as the gusset plates used in mast structures, the recommended uniform pressure impulse over the clearing time (t_c) is given by

$$p_d = \frac{p_r}{2} + p_0 \left[\frac{2U_r}{3U_s} - 0.5 \right]$$

where U_r (in fps) denotes the speed of sound in the reflected region, which is in turn given by

$$U_r = 422 \sqrt{\frac{1.088p_0^2 + 70p_0 + 720}{102.9 + 6p_0}}$$

The duration of the impulse (t_c) is defined as $3D/U_r$ where D is taken as one half of the width of the plate.

For circular cylinders, the pressure impulse representing the diffraction phase pressure loading is given by

$$p_d = \left[\frac{8(p_0 t_T)^{1.13}}{5t_T} \right]$$

where $t_T = D/U$ is the time required for the shock wave to traverse the cylinder diameter. The duration of the diffraction phase pressure for circular cylinders is approximately equal to $5D/U$.

7.5.4 Drag Loading

Drag loading is due to the difference between the stagnation pressure and the local pressure. For incompressible fluids, this difference is simply the dynamic pressure q ; and for compressible fluids it is represented by the so-called impact pressure q_i . For weather-generated winds, when the difference is small, the dynamic and impact pressures are practically equal. However, in the case of air blast, the impact pressure may be considerably greater than the corresponding dynamic pressure, and should therefore be used in calculating drag forces. In this case, the impact pressure is given by

$$q_i(t) = q_{i0} \left[1 - \frac{t}{t_{dd}} \right]^2 e^{-2kt/t_{dd}}$$

where k and t_{dd} were given in the definition of dynamic pressure $q(t)$ above, and q_{i0} denotes the peak impact pressure, given by

$$q_{I0} = q_0 + \left[\frac{q_0^2}{p_0 + p_A} \right]$$

The drag loading on isolated structural elements can be computed as

$$F_{Drag} = C_D \Delta p A$$

where Δp is the difference between the stagnation pressure and the local pressure; A represents the projected area, and C_D is the drag coefficient. In general, the drag coefficient depends on the Reynolds' number (Re) and the Mach number (M). However, it is recommended that for circular members, the drag coefficients be chosen using the following guide:

- $M > 0.48$, $Re > 3 \times 10^5$, $C_D = 0.7$
- $M < 0.48$, $Re > 3 \times 10^5$, $C_D = 0.6$
- $M < 0.48$, $Re < 3 \times 10^5$, $C_D = 1.2$

For weather-generated winds, the Reynolds' number is usually less than 3×10^5 and hence a drag coefficient of 1.2 should be used.

For plates, on the other hand, the drag coefficient is less dependent on Reynolds' number. For plates and cylinder-gusset combinations, drag coefficients should be chosen using the following guide:

- For cylinder-gusset plate combinations: $C_D = 1.25$
- For rectangular plates: $C_D = 1.16$
- For circular plates: $C_D = 1.12$

The approach that must be adopted in computing the blast load on members whose axes are not normal to the blast direction is as follows:

- (a) compute the normal velocity component;
- (b) use the normal velocity to obtain the drag coefficient C_D ;
- (c) compute the force due to the normal velocity, and
- (d) break the normal drag force into global coordinate directions

7.5.5 Interference Effects

Interference effects are not included in the current version of MASTSAS.

7.5.6 Temporal Variation of Blast Load for lattice Masts

The temporal variation of blast loads on components of lattice masts is comprised of the diffraction and dynamic (drag) phases, as shown in Figure 7.13. The members subjected to the blast load are determined based on their orientation with respect to the blast wind

direction. This includes members that are at the “front” and “back” sides with respect to the blast direction, as shown in Figure 7.14.

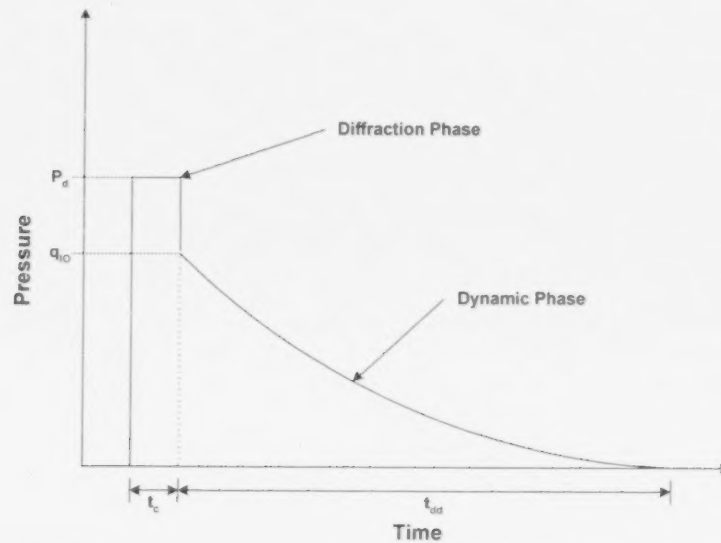


Figure 7.13: Schematic Representation of Temporal Variation of Blast Load for Lattice Mast Members

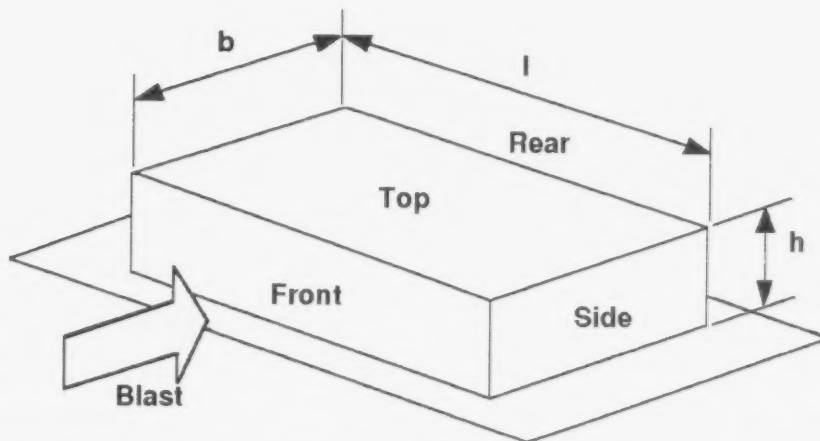


Figure 7.14: Schematic Representation of Structural Regions Affected by Blast Loading

7.5.7 Temporal Variation of Blast Load for Enclosed Masts

The temporal variation of blast loads on components of enclosed masts consist of the diffraction and dynamic phases as shown in Figure 7.15, as suggested by the Lloyd's guide for blast design of ship superstructures.

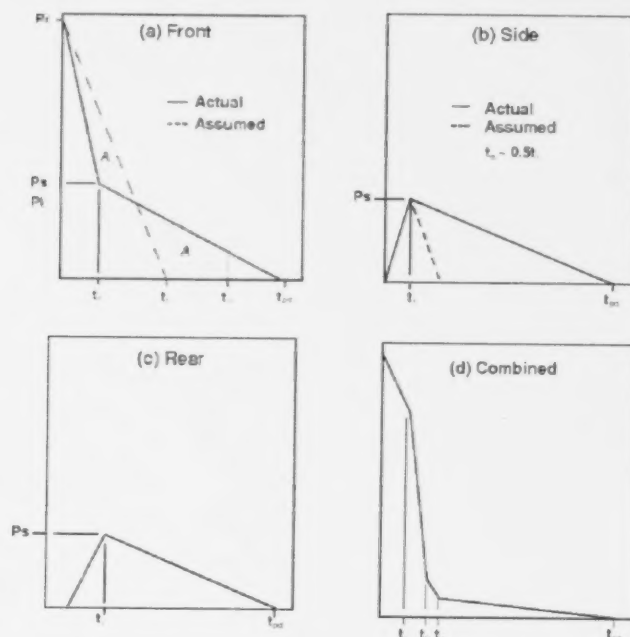


Figure 7.15: Blast Loading on Enclosed Masts (Lloyd's, 2000)

7.6 Underwater Shock Loads (USL)

The forces transmitted to a mast through a ship that is subjected to a shock wave are typically in the form of support accelerations. Underwater shock loads (USL's) can be specified in any the following ways (DMEM 10):

- Shock motion for which the imposed displacements at supports are specified;
- Shock spectrum method (SSM), which requires determination of natural frequencies and mode shapes in order to obtain modal participation factors; and
- Base acceleration method (BAM), which is essentially a static analysis procedure.

For the MASTSAS program, only the SSM and BAM approaches are available to specify the USL's. The information required to define the various types of USL's is presented in the following sections.

7.6.1 Shock Spectrum Method (SSM)

The shock spectrum method is not included in the current version of MASTSAS.

7.6.2 Base Acceleration Method (BAM)

The base acceleration method requires that acceleration components in the vertical, longitudinal, and athwarship directions be provided.

8. FINITE ELEMENT MESH AND ANALYSIS SETUP

The MASTSAS code, like other HOOD applications, uses a *boundary detail* feature for generating finite element models (including loads and boundary conditions). MASTSAS users wishing to create a boundary detail can do so by selecting the “*Boundary Detail*” option provided under the main menu’s “*FE Model*” tab (see Figure 8.1). MASTSAS will then generate a new boundary detail using the default mesh size (see Figure 8.2) and display its property page (see Figure 8.3). Once the boundary detail has been created, users can then create input data for the VAST/DSA finite element system in two steps:

- (1) setting the “*Create Mesh Flag*” option to true and then selecting the “*Apply*” button
- (2) selecting the “*Export to DSA*” button



Figure 8.1: Creating a New Boundary Detail

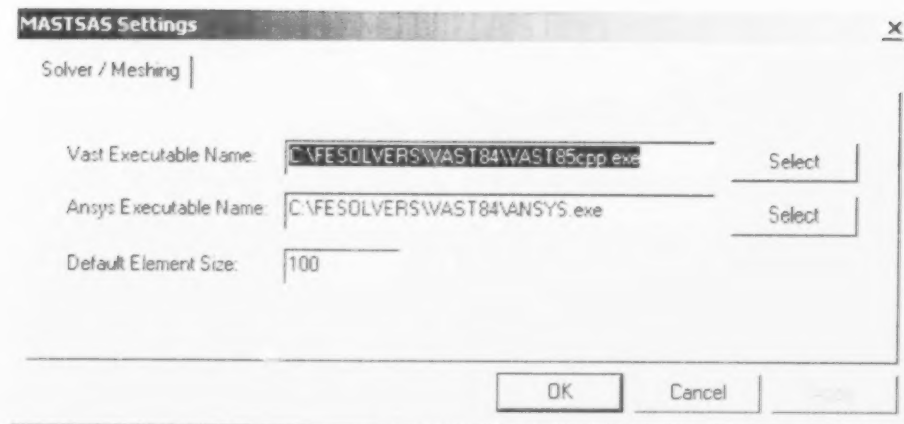


Figure 8.2: Default Mesh Settings GUI

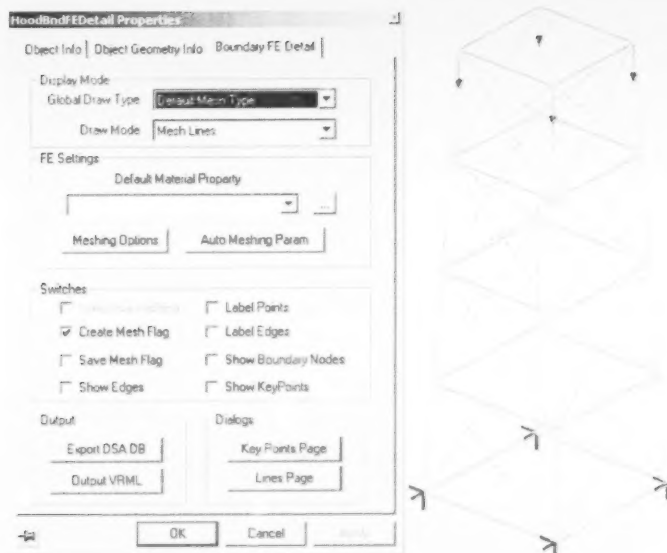


Figure 8.3: Boundary Detail Property Page

Selecting the “*Export to DSA*” option will display an additional dialog box in which users must supply the name of the DSA database to be generated (see Figure 8.4). Users can also create a VAST-compatible GOM file by selecting the “*Create GOM File*” option.

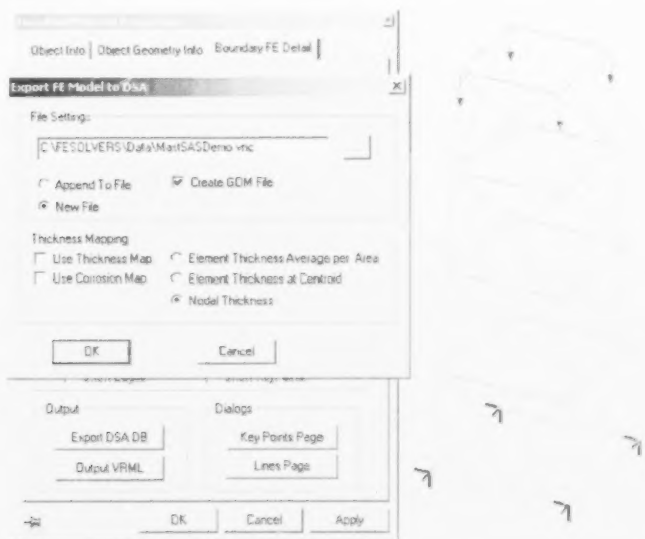


Figure 8.4: Creating a DSA Database of the MASTSAS Model

After creating the DSA/VAST files, users can set up and run a VAST analysis by selecting the "Analysis Setup" option listed under the main menu's "FE Model" tab (recall Figure 8.1). In order to automate the procedure for running the finite element solver (currently only the VAST system is supported), MASTSAS provides a simple dialog box (see Figure 8.5) which allows users to select the desired analysis type to be performed and launch the finite element solver.

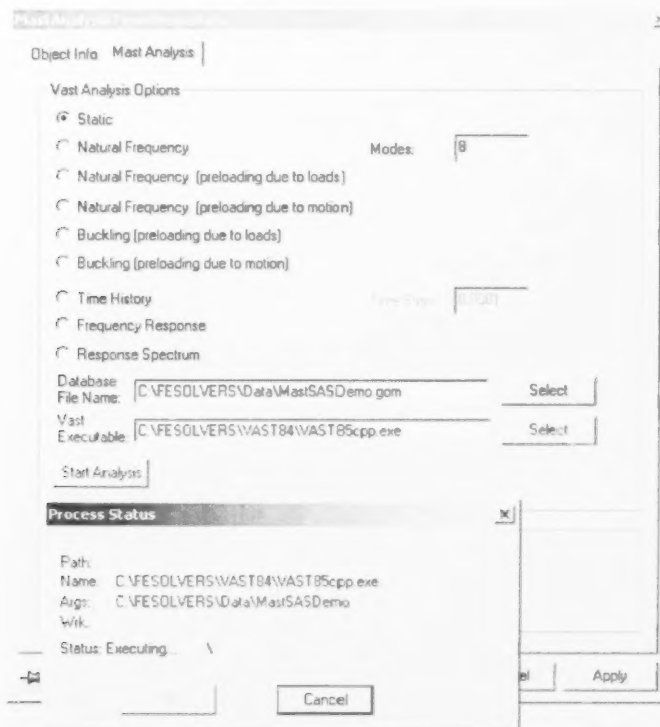


Figure 8.5: Input Data for Analysis Setup

9. POST-PROCESSING

The MASTSAS system is designed to provide post-processing capabilities that are specific to mast structural systems. General finite element post-processing capabilities, such as display of displacements, stresses, strains, mode shapes, etc., can be done within the supported finite element solver packages (i.e., DSA-VAST and ANSYS, respectively). With this premise, MASTSAS' post-processing capabilities are required only for lattice type masts, whose members require additional integrity checks. The MASTSAS system provides post-processing capabilities for evaluating the adequacy of:

- (a) the strength of lattice mast members;
- (b) the stability of lattice mast members; and
- (c) the ability of lattice mast members to avoid vortex shedding.

9.1 Strength and Stability Adequacy Checks

The procedures for providing these adequacy checks are in accordance with the methodologies described by Crocker et al (1999). The steps involved in evaluating the strength and stability requirements of the mast members are summarized in Figure 9.1 and Figure 9.2.

In accordance with the approach adopted by Crocker et al (1999), the unsupported length of any given member is taken as the distance from joint to joint, without consideration of the reduction in member length due to connecting gusset plates. Also, the effective length factor for each mast member is assumed to be 0.9.

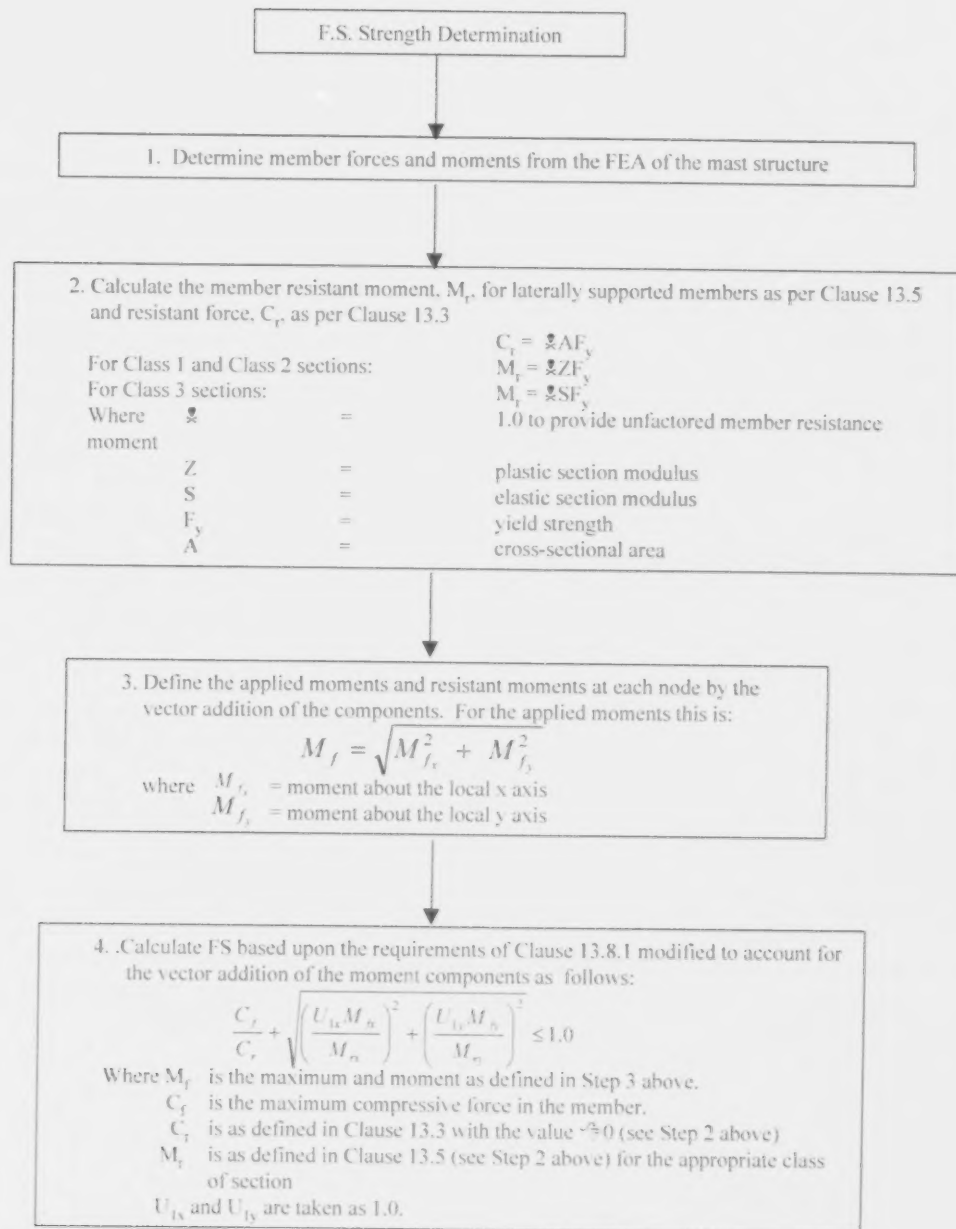


Figure 9.1: Flowchart of Algorithm to Determine the Mast Members Factor of Safety (Crocker et al, 1999)

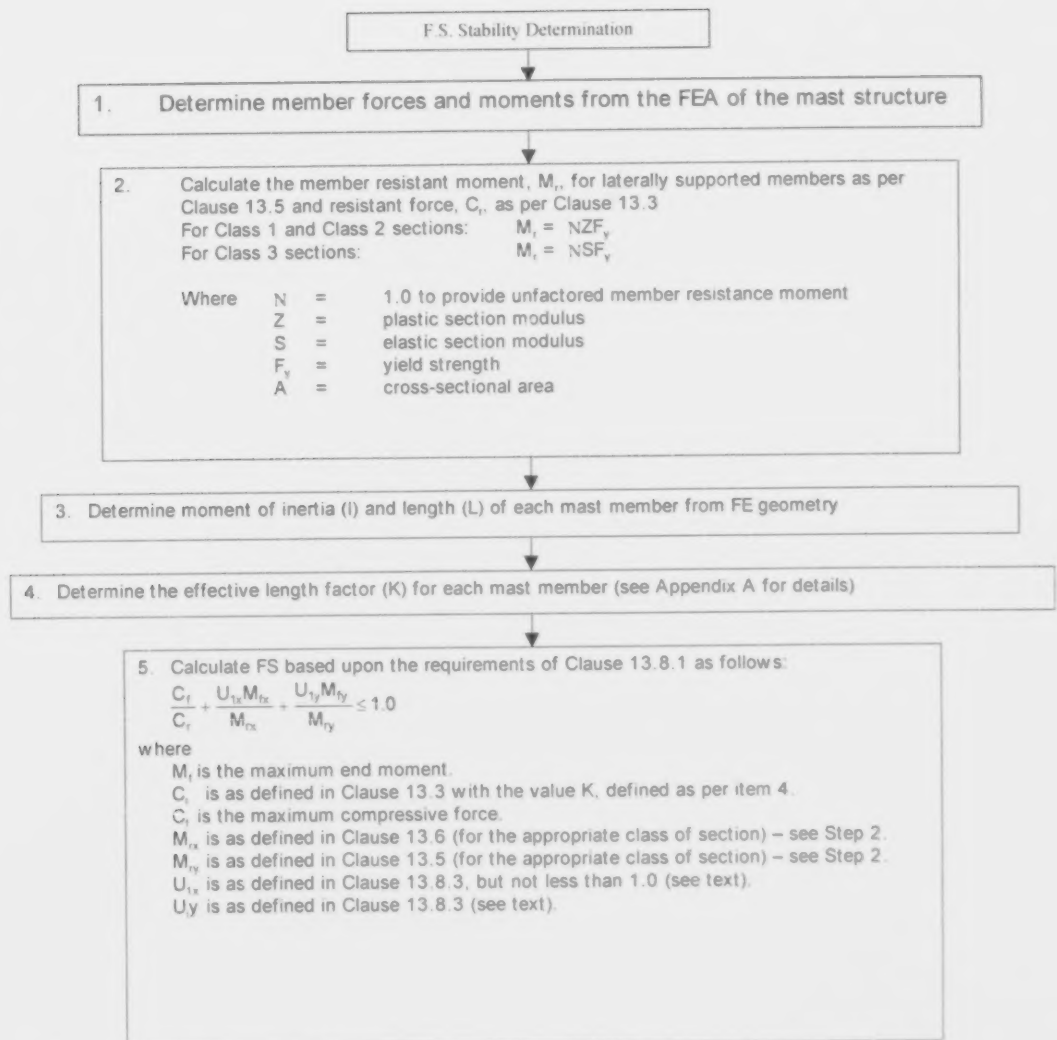


Figure 9.2: Flowchart of Algorithm to Determine the Mast Members Factor of Safety (Crocker et al, 1999)

9.2 Vortex Shedding Checks

The procedures for checking the adequacy of members against vortex shedding are carried out in accordance with the approach described by Norwood (1977). When wind blows across slender prismatic or cylindrical members, vortices can be shed alternatively from one side to the other, giving rise to fluctuating forces at right angles to the wind direction along the length of the body. Checks are required to ensure that the mast does not contain members whose natural frequencies are in resonance with the frequency of vortex shedding and of the force fluctuations.

The frequency of vortex shedding and of the force fluctuations is given by the relation

$$f_v = \frac{S_n V}{D}$$

where f_v denotes the frequency in Hertz (cycles/sec), V represents the mean wind speed, D is the diameter; and S_n is the Strouhal number, which depends on (i) the mast member's cross-sectional shape, and (ii) the Reynolds' number (for circular sections) ($Re = 0.0625 VD \times 10^5$). Typical values of the Strouhal number S_n are provided below:

- Circular, $Re < 2 \times 10^5$ $S_n = 0.18$
- Circular, $Re > 2 \times 10^5$ $S_n = 0.25$
- Others $S_n = 0.15$

The natural frequency of a mast member vibrating in a plane normal to the wind direction is given by

$$f_m = \frac{c}{2\pi} \sqrt{\frac{gEI}{wL^4}}$$

where f_m represents the natural frequency of the member in Hertz (cycles/sec); g is the acceleration due to gravity; E is the Young's modulus; I denotes the moment of inertia; w represents the mass per unit length; L is the length of the member; and c is a coefficient which depends on the end conditions of the member, as defined below:

- Pinned-pinned: $c = 9.87$;
- Clamped-clamped: $c = 22.4$; and
- Clamped-pinned: $c = 15.4$

The wind velocity at which resonance occurs is obtained by simply equating f_v and f_m , such that the resonant velocity V_r is given by

$$V_r = \frac{Dc}{2\pi S_n} \sqrt{\frac{gEI}{wL^4}}$$

To avoid vortex shedding in a member, it is desirable that its resonant velocity V_r be higher than that of the design wind load.

10. REFERENCES

- [1] Lichodziejewski, M., MacAdam, T., "HOOD User's Manual", Martec Technical Report TR-99-05, February, 1999
- [2] Norwood, M.E., "Engineering Standard for the Design of Lattice Masts of Ships, Vol. 1: Design Standard", Martec Report, 1977.
- [3] Crocker, E.J., Delbridge, G. and Kumar, R., "Development of HyperVAST Model for IRO Mast – Phase I", Martec Report No.: TR-98-13, 1998

DOCUMENT CONTROL DATA		
(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)		
1. ORIGINATOR (the name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Centre sponsoring a contractor's report, or tasking agency, are entered in section 8.)	2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable).	
Martec Limited, Suite 400, 1888 Brunswick St., Halifax, NS, B3J 3J8	UNCLASSIFIED	
3. TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title).		
MASTSAS – Mast Structural Analysis Software, Reference Manual		
4. AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)		
Koko, T. S., Brennan, D. P., Norwood, M. E., MacAdam, T., and Rushton, P. A.		
5. DATE OF PUBLICATION (month and year of publication of document)	6a. NO. OF PAGES (total containing information include Annexes, Appendices, etc.)	6b. NO. OF REFS (total cited in document)
April 2004	82 (approx.)	3
7. DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered).		
CONTRACT REPORT		
8. SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include address).		
Defence R&D Canada – Atlantic PO Box 1012 Dartmouth, NS, Canada B2Y 3Z7		
9a. PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant).	9b. CONTRACT NO. (if appropriate, the applicable number under which the document was written).	
11gc26	W7707-008062/001/HAL	
10a. ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document.)	10b. OTHER DOCUMENT NOS. (Any other numbers which may be assigned this document either by the originator or by the sponsor.)	
Martec TR-02-26	DRDC Atlantic CR 2004-088	
11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)		
☒ (x) Unlimited distribution ☐ () Defence departments and defence contractors; further distribution only as approved ☐ () Defence departments and Canadian defence contractors; further distribution only as approved ☐ () Government departments and agencies; further distribution only as approved ☐ () Defence departments; further distribution only as approved ☐ () Other (please specify):		
12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in (11)) is possible, a wider announcement audience may be selected).		

13. **ABSTRACT** (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual)

MASTSAS (MAST Structural Analysis Software) is a special-purpose, PC based computer program for rapid finite element modeling of warship mast structures. The MASTSAS software was developed under contract to DRDC Atlantic, using its HOOD (Hierarchical Object Oriented Database) toolkit. This Reference Manual, one of a series of three manuals documenting the MASTSAS software, provides the theoretical foundations and details on the capabilities offered by the program.

14. **KEYWORDS, DESCRIPTORS or IDENTIFIERS** (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus, e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title).

Mast
Ship structure
Finite element modeling
Composite structure
HOOD
Object oriented programming